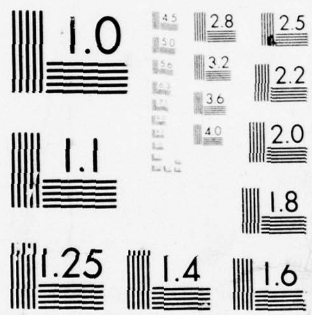
AD-A043 448 NATIONAL ACADEMY OF SCIENCES-NATIONAL RESEARCH COUNCI--ETC F/G 20/4
PROCEEDINGS OF THE WORKSHOP ON NUMERICAL HYDRODYNAMICS.(U)
JUL 75 N00014-67-A-0244-0013

UNCLASSIFIED NL

1 OF 5
AD
A043 448



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

0
B.S.

Proceedings of
The Workshop on

Numerical Hydrodynamics

The National Research Council
INSTITUTE OF MEDICINE · NATIONAL ACADEMY OF ENGINEERING · NATIONAL ACADEMY OF SCIENCES

Sponsored by the

Committee on Undersea Warfare
National Research Council

and the

Fluid Dynamics Branch
Office of Naval Research

NATIONAL ACADEMY OF SCIENCES

BIBLIOGRAPHIC DATA SHEET	1. Report No.	2.	3. Recipient's Accession No.
4. Title and Subtitle Proceedings of the Workshop on Numerical Hydrodynamics			5. Report Date July 1975
			6.
7. Author(s) Committee on Undersea Warfare and Fluid Dynamics Branch, Office of Naval Research			8. Performing Organization Rept. No.
9. Performing Organization Name and Address National Research Council National Academy of Sciences 2101 Constitution Avenue, NW Washington, D.C. 20418			10. Project/Task/Work Unit No. 11. Contract/Grant No. N00014-67-A-0244-0013
12. Sponsoring Organization Name and Address Fluid Dynamics Branch Office of Naval Research 800 North Quincy Street Arlington, VA 22217			13. Type of Report & Period Covered Proceedings of a Workshop 14.
15. Supplementary Notes			

16. Abstracts

The potential of numerical fluid dynamics as a design tool for naval architects and hydrodynamicists is explored both in formal presentations and in informal discussion. The status of ship design and the theory and procedures of fluid dynamics are discussed in the light of the present and potential status of numerical hydrodynamics and requisite computer capability. Conclusions are drawn regarding promising avenues of further research.

17. Key Words and Document Analysis. 17a. Descriptors

Fluid Dynamics
Hydrodynamics
Numerical Hydrodynamics
Computer Technology
Computational Fluid Dynamics
Naval Architecture

17b. Identifiers/Open-Ended Terms

17c. COSATI Field/Group

18. Availability Statement	19. Security Class (This Report) UNCLASSIFIED	21. No. of Pages 333
Distribution Unlimited	20. Security Class (This Page) UNCLASSIFIED	22. Price

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) (6) Proceedings of the Workshop on Numerical Hydrodynamics.		5. TYPE OF REPORT & PERIOD COVERED Proceedings of a Workshop
		6. PERFORMING ORG. REPORT NUMBER (15)
7. AUTHOR(s) Committee on Undersea Warfare and Fluid Dynamics Branch, Office of Naval Research		8. CONTRACT OR GRANT NUMBER(s) N00014-67-A-0244-0013
9. PERFORMING ORGANIZATION NAME AND ADDRESS National Research Council National Academy of Sciences 2101 Constitution Avenue Washington, D.C. 20418		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS
11. CONTROLLING OFFICE NAME AND ADDRESS	(11)	12. REPORT DATE July 1975
		13. NUMBER OF PAGES 342
14. MONITORING AGENCY NAME & ADDRESS(if different from Controlling Office) (12) 335p.		15. SECURITY CLASS. (of this report) UNCLASSIFIED
		15a. DECLASSIFICATION DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

Distribution Unlimited

DISTRIBUTION STATEMENT A
Approved for public release;
Distribution Unlimited

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

Fluid Dynamics
Hydrodynamics
Numerical Hydrodynamics
Computer Technology
Computational Fluid Dynamics
Naval Architecture

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

The potential of numerical fluid dynamics as a design tool for naval architects and hydrodynamicists is explored both in formal presentations and in informal discussion. The status of ship design and the theory and procedures of fluid dynamics are discussed in the light of the present and potential status of numerical hydrodynamics and requisite computer capability. Conclusions are drawn regarding promising avenues of further research.

DD FORM 1 JAN 73 1473 EDITION OF 1 NOV 65 IS OBSOLETE

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

092250

1B

Proceedings of the Workshop on

NUMERICAL HYDRODYNAMICS

Held at the
National Academy of Sciences
May 20-21, 1974

Sponsored by the
Committee on Undersea Warfare
Assembly of Mathematical and Physical Sciences
National Research Council

and the
Fluid Dynamics Branch
Office of Naval Research

NATIONAL ACADEMY OF SCIENCES
Washington, D.C. 1975

NOTICE

The project which is the subject of this report was approved by the Governing Board of the National Research Council, acting in behalf of the National Academy of Sciences. Such approval reflects the Board's judgment that the project is of national importance and appropriate with respect to both the purposes and resources of the National Research Council.

The members of the committee selected to undertake this project and prepare this report were chosen for recognized scholarly competence and with due consideration for the balance of disciplines appropriate to the project. Responsibility for the detailed aspects of this report rests with that committee.

Each report issuing from a study committee of the National Research Council is reviewed by an independent group of qualified individuals according to procedures established and monitored by the Report Review Committee of the National Academy of Sciences. Distribution of the report is approved, by the President of the Academy, upon satisfactory completion of the review process.

PREFACE

On November 12, 1973 representatives of the Office of Naval Research met with the Committee on Undersea Warfare of the National Academy of Sciences to discuss the possibility of a jointly sponsored Workshop on Numerical Hydrodynamics. The objective of the Workshop, as proposed to the Committee, would be to assess the potential of numerical fluid dynamics as a design tool for hydrodynamicists and naval architects, and to identify profitable areas of further research in this field. The Committee agreed with both the importance and timeliness of the proposed Workshop, believed that the objectives could be realized through formalized discussion and debate between hydrodynamicists, naval architects, mathematicians and computer specialists, and recognized the value of a published proceedings to the available literature on fluid dynamics. The Committee, therefore, joined with the Office of Naval Research in sponsoring the Workshop subsequently held at the National Academy of Sciences on May 20-21, 1974.

As an initial step in planning the Workshop a Program Committee was established under the Chairmanship of Dr. George F. Carrier of Harvard University. Dr. Robert W. Buchheim, Chairman of the Committee on Undersea Warfare, Dr. J. P. Breslin, Stevens Institute of Technology and Dr. Garrett Birkhoff of Harvard University served as members of the Committee. Meeting on February 28, 1974, the Committee developed the basic plans for the Workshop, identified the topics for formal presentation, and indicated a preferred scheduling of sessions and events. At this meeting Dr. Carrier agreed to serve as Chairman of the Workshop.

A subsequent step in planning the Workshop was to establish a Review Committee with the assigned task of developing a critique of the Workshop, and of indicating collective judgment as to the pace and direction for further research in numerical hydrodynamics. Chaired by Dr. Carrier, the Review Committee was comprised of Dr. J. P. Breslin,

Stevens Institute of Technology, Mr. Marshall P. Tulin, Hydronautics, Inc., Dr. Garrett Birkhoff, Harvard University, Dr. Thomas D. Taylor, the Aerospace Corporation, and Dr. Steven A. Piacsek, Naval Research Laboratory. The report of the Review Committee is included in the Proceedings as the final document.

Of the eight formal presentations at the Workshop, only six manuscripts were submitted for inclusion in the Proceedings. These six papers are included here, along with an edited selection of the discussions which followed their presentation. Also included are three informal contributions which added significantly to the exchange of information during the Workshop.

The Office of Naval Research and the National Academy of Sciences-National Research Council deeply appreciate the time, effort and thoughtful contributions of those who participated in the Workshop, and those who served on the Program and Review Committees.

WORKSHOP PROGRAM COMMITTEE

Professor George F. Carrier, Chairman
Harvard University

Dr. J. P. Breslin
Stevens Institute of Technology

Dr. Robert W. Buchheim
Chairman, CUW

Dr. Garrett Birkhoff
Harvard University

WORKSHOP REVIEW COMMITTEE

Professor George F. Carrier, Chairman
Harvard University

Dr. Steve A. Piacsek
Naval Research Laboratory

Dr. Garrett Birkhoff
Harvard University

Dr. Thomas D. Taylor
The Aerospace Corporation

Dr. J. P. Breslin
Stevens Institute of Technology

Mr. Marshall P. Tulin
Hydronautics, Inc.

ONR REPRESENTATIVES

Mr. Ralph Cooper

Mr. Morton Cooper

NAS-NRC REPRESENTATIVE

Mr. Lee M. Hunt

TABLE OF CONTENTS

PROGRAM

Workshop on Numerical Hydrodynamics
May 20-21, 1974

Chairman's Opening Remarks
George F. Carrier, Harvard

Welcome Address
Captain F.H. Featherston, USN, ONR

SESSION I: HYDRODYNAMICS PROBLEMS

Moderator: M.P. Tulin, Hydronautics

Ship Performance Prediction and Numerical Hydrodynamics: An Opportunity -- W.E. Cummins, NSRDC

The Navy's Hydrodynamics Problems Now and in the Future -- A Designer's View -- R.S. Johnson & Peter A. Gale, NAVSEC

Selected Problems in Undersea Technology -- R. Hogland, ARPA
(Not included in Proceedings)

SESSION II: NUMERICAL HYDRODYNAMICS

Moderator: T. Taylor, Aerospace

Numerical Hydrodynamics as a Mathematical Science -- G. Birkhoff, Harvard

Numerical Hydrodynamics -- Present and Potential -- C.W. Hirt, Los Alamos

Progress in Computational Fluid Dynamics at NASA Ames Research Center -- R.W. MacCormack, NASA Ames

Numerical Simulation of Turbulent Flow -- Steven A. Orszag, MIT

SESSION III: COMPUTER CAPABILITY

Moderator: S. Piacsek, NRL

Future Directions in Computers -- M.L. Dertouzos, MIT (Not included in Proceedings)

Some Aspects of Computing with Array Processors -- C.E. Leith, NCAR

INFORMAL CONTRIBUTIONS

A Comment on the Efficiency of Finite Difference Methods for Fluid Dynamics -- Hans Lugt, NSRDC

Non-Linear Numerical Solutions of Two-Dimensional Wave Problems -- Nils Salvesen, NSRDC

Research on Hydrodynamic Free Surfaces -- Joel C.W. Rogers, NOL

SESSION I

HYDRODYNAMICS PROBLEMS

INTRODUCTORY REMARKS

Captain F. H. Featherston, USN
Deputy and Assistant Chief
Office of Naval Research
Arlington, Virginia

I would like to welcome you on behalf of the Office of Naval Research. ONR is the premier organization in the area of innovative and forward-looking research. You will note on the program that Admiral Van Orden, the Chief of Naval Research, was scheduled to open the morning session. Unfortunately his time was unexpectedly preempted. He has asked me, as his deputy, to stand in for him and to offer his apologies of our Chief Scientist, Dr. William P. Raney, who was called to the same meeting. He hopes to be with you before the end of the Workshop.

The Office of Naval Research, which our Chief Scientist likes to refer to as a "wild card", takes particular pride in being the first organization of its type to formally serve as an interface between government and the civilian scientific and engineering communities. Therefore, I think it particularly appropriate that this Workshop should be jointly sponsored by, and housed within, an organization which, for a longer period of time, has so admirably served a similar interface function. Adjoining us, in the Board Room, there hangs above the fireplace a painting of President Abraham Lincoln flanked by a number of representatives from government and the scientific community. The painting commemorates the signing, in 1863, of the Congressional Charter which established the National Academy of Sciences. One of the representatives who attended President Lincoln on that occasion was a Commander in the U.S. Navy. I point this out

PRECEDING PAGE BLANK - FI

in order to further emphasize the Navy's long interest in and support of the scientific endeavor, and as further witness to the similarity of the goals and functions of the two sponsoring organizations.

The location of the Workshop also brings to mind the ghostly presence of the old Main Navy and Munitions buildings which used to sit across the street. Before and during World War II both NACA and the Bureau of Aeronautics were housed in these buildings. I like to think that a great deal of the success we had in meeting the wartime technical challenges imposed on naval aviation can be attributed to the close proximity of these two organizations, and the ease with which communication could be carried out at the working level. The preeminence achieved by naval aviation during this period provided momentum which carried well into the post-war years, and had a significant impact on aircraft design even in the other Services.

I have introduced this bit of naval aviation history for two reasons: (1) Some of the technical problems, particularly those of aerodynamics, are similar to the issues to be discussed during your meeting, and (2) by this assembly of experts from across the country we hope to recreate, albeit fleetingly, the type of informal dialogue which proved so productive and stimulating in the past. Specifically, we hope that in the next two days you will be able to define what numerical hydrodynamics can do for the Navy, especially in the design of more efficient surface ships and craft.

You are all familiar with the desire, indeed the need to build a modern, economic Navy, and with the resulting emphasis being placed on such approaches as hi-low mixes, catamarans, hydrofoils, surface effect ships, and with our emphasis on innovative approaches to the overall objective are formidable; a fact which further emphasizes the importance and the timeliness of your meeting. Hopefully, our older, cut-and-try methods of achieving the desired capability in ship performance

can be supplemented by applications of the computational sophistication which has characterized the one fluid domain of the aircraft. The application of numerics to hydrodynamics is likely to be more difficult, but, by the same token, the results, if successful, may be of far greater importance to those vehicles which ride the interface of a two-fluid domain. Engineering advances such as the super cavitating propeller and the use of polymer additives to reduce hull friction are highly significant and will, undoubtedly, continue to emerge. But, if we are to build the Navy we must have by the year 2000, we need advances across the entire spectrum of disciplines and sub-disciplines which contribute to improvements in ship design. Improvements in our understanding of hydrodynamic processes, as well as in their calculation and prediction, is certainly one of the more critical lines in that spectrum. We look to you, the researchers and practitioners in the fields of naval architecture, hydrodynamics, mathematics and computers, to tell us what must be done to transform numerical hydrodynamics into the powerful tool it promises to become.

On behalf of Admiral Van Orden I wish you a most stimulating and productive meeting. Thank you.

SHIP PERFORMANCE PREDICTION AND NUMERICAL HYDRODYNAMICS: AN OPPORTUNITY

W. E. Cummins
Head, Ship Performance Department
Naval Ship Research and Development Center
Bethesda, Maryland

INTRODUCTION

This paper is a review of the unsolved problems of ship performance prediction, presented with the hope that the newly developing techniques of numerical hydrodynamics can contribute to the solution of some or all. As I am merely offering opportunities, not promises of success, my task is not difficult. Surely, some of these problems will remain with us for a long time. But considering the sometimes remarkable ingenuity of the workers in the towing tanks, coupled with the brilliance of the investigators of computer science, we can anticipate that a properly organized and adequately funded effort will make advances which will significantly increase our capabilities in performance prediction, and consequently improve the design process which depends on these predictions.

Ever since Froude started his experiments with ship models about a century ago, towing tanks have been predicting the hydrodynamic performance of ships as part of the design process. In view of the continually growing number of towing tanks, and the ever increasing effort devoted to making predictions and expanding the technology of prediction, we must be doing something right. As a matter of fact, there have been a number of achievements during the past century of which we are proud. And the frequency of these achievements is increasing, in roughly the same exponential growth pattern as the rest of modern technology. The progress is attributable not only to improved experimental techniques

Preceding Page Blank

but as well to an increasing use of theory in the solution of practical problems. Today, quite sophisticated techniques based on hydrodynamic theory, statistical theory, and classical mathematics as well as computer science are used in the solution of design problems, supplementing and sometimes replacing the physical model. We attack problems today which would have been idealized to the point of absurdity only a few years ago. In many cases we have achieved a degree of confirmed precision which reduces the dependence on the engineering intuition which is required in seeking solutions by trial and error.

Why then, do we need the help of the new techniques of numerical hydrodynamics? The answer, as hopefully will be made clear, is that there is a sort of barrier which these current techniques, powerful though they are, show no signs of passing. At the same time, it hardly seems profitable to use the probably costly procedures of numerical hydrodynamics to solve problems which the traditional methods have handled well. I would expect that we will ultimately arrive at a mixed strategy, in which we use *both* approaches in a coordinated fashion, achieving results far beyond the capability of either.

WHY PERFORMANCE PREDICTION?

The functions of the towing tanks are fairly well known. However, in order to place the unsolved problems in proper perspective, it is useful to review the function of the towing tank in some detail, with an indication of what we do well and where there are difficulties. The Ship Performance Department of the Naval Ship Research and Development Center is not completely typical, but its mission is well suited for this purpose as we participate in a continuing dialog with the designers, operators, and managers in the Navy.

The oldest function of the towing tank, and the one which brought them into existence, is the prediction of the powering requirements of

a ship design, and supplying to the designer information which makes it possible to have a three-way match of the hull, the propeller, and the power plant. This requires an accurate estimate of the resistance of the ship, the characterization of the propeller, and finally the estimate of shaft horsepower and RPM as a function of speed. The techniques used are essentially those developed by Froude, Taylor, and others many years ago. They make use of ship and propeller models, and extrapolation to full scale is complicated by a mix of Froude and Reynolds' effects. It is well known that these effects are not truly separable.[1] Nevertheless any competent and experienced towing tank organization, working with hull forms of "conventional" type, military or commercial, consistently makes predictions which trials confirm to be accurate within five percent on power, and even better on RPM. But, as will be shown below, there are problems when hull forms depart from the conventional.

Related to this function is the recently assigned responsibility of the Ship Performance Department for propeller design. The objective is a highly efficient propeller, matched to the wake in which it operates, and which excites little vibration. The geometries of these propellers are sometimes highly complex, and their design is carried out by means of computer programs developed from classical hydrodymanic theory. The methods work, though there are problems in estimating loads in off design conditions.

Another closely related function is the development of the hull lines, using hydrodynamic singularities and linear wave theory. The success of these efforts is sometimes spectacular, particularly for ship types of unusual form, where historical data is sparse or non-existent. The Catamaran is an example.

After almost a century of powering prediction, the towing tanks have been given the additional task of predicting dynamic performance:

maneuvering and seakeeping. The seakeeping work in particular has received emphasis during the last two decades, following the appearance of a suitable statistical model of the seaway.[2] Theoretical and experimental procedures for prediction of seakeeping performance have become standard tools. There are versions available for use in early hull form development as well as final evaluation. But since this is a frontier area, there remain many problems.

Of particular note in this area of dynamics is the characterization of submarine maneuvering ability. The standard technique is the use of a physical model to obtain the coefficients to insert in a mathematical model. Simulation then provides a means of judging the adequacy of the control system and provides a means for establishing operational strategies. An example of the latter is the specification of the procedures to be followed in case of a control system failure for each submarine in the fleet. In view of the high speed of modern submarines and the small range of depths in which they can safely operate, the responsibility for proper performance prediction is high. The tools have not always been sufficient for the needs.

A final area of responsibility of the towing tank is of the utmost importance, but cannot be spelled out with any precision. It can be vaguely described as the evaluation of the adequacy of the hydrodynamic design of all subsystems of the ship. The customer pays little attention to this aspect until a ship exhibits some major or minor flaw, and then he complains loudly if we have not predicted the problem. Examples of conditions to be anticipated are: unexpected cavitation on the hull or an appendage, poor flow around hull openings, separation and vortex shedding, bubbles which interfere with sonar operation, propeller blade erosion, and any steady state or dynamic phenomenon which impairs the operational performance of the ship.

All of the above discussion has been given in terms of conventional design support for conventional ships. It applies equally well

to the so-called high performance ships in which the Navy now has an interest. The difference is that we have little experience and inadequate facilities for treating the problems at the very high speeds anticipated for some of these craft. And yet it is obvious, in practically every aspect, that the potential consequences of an error are greatly magnified. For an ordinary ship, with just a little care you can be sure it will float upright. Turn the propeller, and it will move. Turn the rudder, and it will move in a curved path without great danger. If not properly and precisely designed, it may not meet specifications, but it is rare that the ship will be a complete failure. But for high performance craft the margin for error is drastically reduced. The trade-offs are no longer carried out among a large family of more or less acceptable solutions, but rather as a search for even one solution which is practically feasible. And the technology base is both meager and costly to expand.

THE ROLE OF NUMERICAL HYDRODYNAMICS

A review of the problem areas of the previous section suggests that they can be roughly divided into two categories -- the "global", in which we are concerned with some aspect of performance of the entire ship in its environment, and the "local", in which we are concerned with the nature of the flow and its physical consequences in a small region. A "local" phenomenon might have "global" impact, but its definition need not extend beyond a limited and fairly well defined region. The prediction of resistance and powering is global. But the performance of a propeller in the ship's wake is local. The prediction of ship behavior in a storm sea is global. The time history of a slam is local.

It is my belief that the greatest opportunities for numerical hydrodynamics lie with the local problems. With some exceptions, the global problems would place such enormous demands for computer time

that economics alone would dictate that other approaches be followed. This is particularly true of ship dynamics, where gross statistical behavior is the usual objective. On the other hand, local problems can frequently be constructed in such a way that they are limited in both time and space, and thus would encourage the finite step approaches that are typical of numerics.

Further, the greatest opportunities for these methods would seem to be where the traditional or classical methods have experienced difficulty. The older methods have been most successful when various effects in a complex phenomenon could be separated and examined separately, thus providing a sort of linearization, even if only approximate. Typical examples are the separation of frictional and residual (or "wave") resistance, and the characterization of ship seakeeping qualities in a real sea as the sum of the behavior in an infinite number of regular waves. Another sort of resolution is provided by "strip theory" which breaks the three dimensional flow relative to a ship into a set of flows about the transverse sections which can be analyzed by two dimensional theory. But when the physical phenomenon cannot be broken down in some such fashion, classical theory has generally failed, and the only alternative has been a limited and frequently expensive experiment. There is little opportunity to examine sensitivity to variations in design, and guidance provided to the designer is far from adequate for his needs.

Thus, I would conclude that the greatest opportunities for these approaches are the solution of local problems which are inherently nonlinear, strongly three dimensional, or both. Obvious candidates are cases in which some major viscous phenomenon is a factor (such as transition, separation, vortex shedding, etc.) and free surface or cavity flows (such as steep waves, cavitation, ventilation, cavity collapse, etc.).

A CATALOG OF PROBLEMS

Global Problems

Even though the point has just been made that "global" problems offer few opportunities for numerical hydrodynamics, this catalog will start with several such problems which have resisted the classical approaches, and which urgently need solution. All are strongly viscous in their nature, and strongly three dimensional as well. It may be that a feasible attack would require that they be broken up into sets of local problems, but if such an approach is used, it is important to remember that the objective is a "total" solution.

The first is a steady state problem: the lift on a body of revolution, either pure or with appendages, at an angle of attack, (Figure 1). The nature of the flow is understood in a very rough sense, but there is no usable theory, in spite of a number of attempts. We are very much dependent upon model tests to obtain the coefficients in the equations of motion. Since viscous effects, particularly separation and vortex shedding play an essential role, the possibility of scale effects is high. We have no real handle on this one.

The second is also a submarine problem, but is transient in its nature. The operational problem is the behavior of a submarine in a high speed turn. Two things happen which are not accurately predicted by simulation of the equations of motion -- a large transient roll ("snap roll") and a sometimes large change in depth of submergence. The flow is complex, and there are many possibilities for local flows to trigger large effects.

The next problem takes us back to the area explored by Froude many years ago -- the prediction of ship powering -- but now for the large, full form vessels which have become typical in many modern applications. At the model end of the problem there are questions about

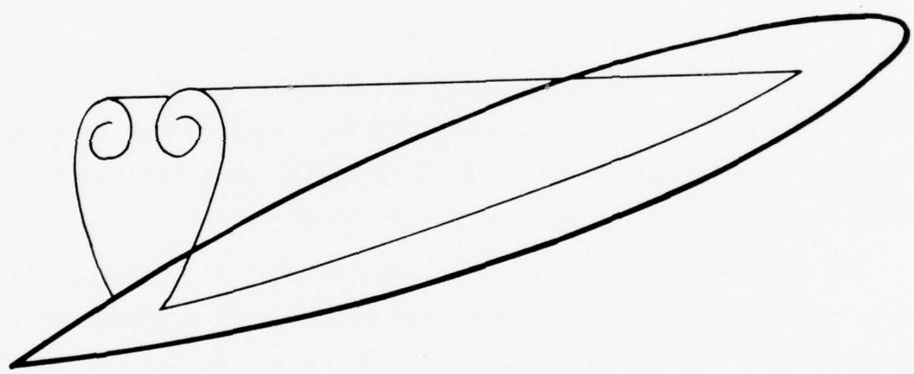


Figure 1 - The Lifting Body

the location of transition on these full bows, and the possibility of separation in the stern region. This creates doubt about the usual procedure of separating frictional and wave resistance. Spray drag is also sometimes large. Scale effects may be so critical that they may invalidate the model experiment. In order to get any correlation at all, some investigators have had to introduce large, arbitrary "corrections" to their predictions, sometimes as much as 25 percent.

The final problem is one of ship motion. The general successes with strip theory for predicting response functions to regular waves are matched by one spectacular failure -- the estimation of the response of a hull form with a large bow bulb or dome.[3] The predictions are so far off as to be useless. Both viscous separation and three dimensional effects are suspected to be contributors. In any case, the responses are strongly non-linear, which is most surprising, considering that for ordinary hull forms linearity is a still rather good approximation when the ship is slamming and taking green water over its decks.

Perhaps it is too much to hope for the early solution of any of the above problems, but an attack which could add to our insight into any of them would be of considerable value.

Free Surface Problems

We begin our discussion of local problems with a group of free surface problems, with "free surface" taken in its most general sense, a boundary between two fluids whose position is not physically fixed. Because these problems are local, they are more precisely definable than the previous group. Further, they may be placed in a rough order of increasing complexity, and a solution to any of them might be an important step in the solution of problems further down the list.

We start with the simple two dimensional planing surface, see Figure 2. While this problem is not of great significance in itself --

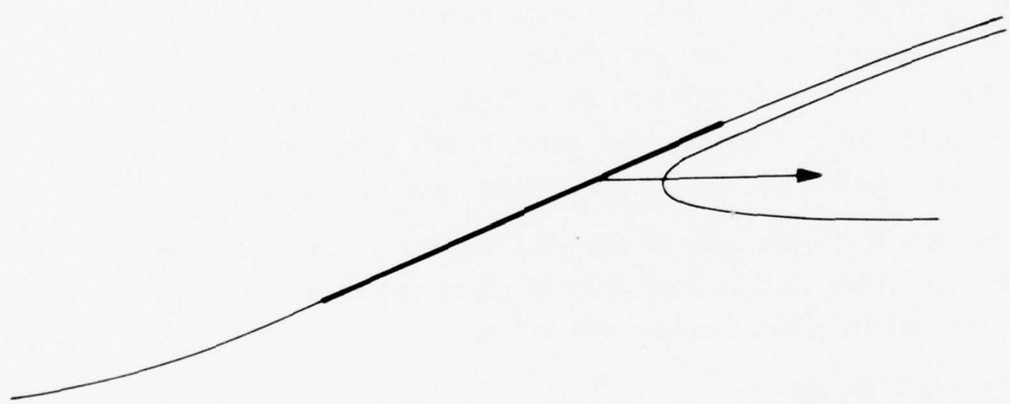


Figure 2 - Planing Surface

a theory exists[4] -- it is an important challenge and first step for numerical hydrodynamics into the domain of the non-linear free surface. There is the problem of the shape of the curving free surface ahead of the body, as well as the wave train behind it. And there is the need for disposing of the "spray" which is shot forward and in a gravity field will reenter the flow.

Now suppose the planing surface to be subjected to a vertical oscillation, Figure 3. What is the time history of the pressure on the surface? Already, we are beyond the range of existing theory or data, and a solution could provide useful insights. This problem is also a useful stepping stone into the unsteady, non-linear free surface domain.

Now consider a two dimensional wedge dropping onto the free surface, Figure 4. If we examine the flow on one side of the plane of symmetry and introduce a moving system of axes, we have an obvious similarity to time dependent planing flow. Two dimensional wedges have been used to study slamming, but the time histories of the theoretically predicted pressures have not correlated very well with experiment. Numerical hydrodynamics could provide an important breakthrough.

Now consider a real slam, in which the forward portion of the ship has emerged from the water and is reentering, Figure 5a. The usual treatment of this problem has been to use a sort of strip theory, looking at the reentry, section by section, and treating them as two dimensional wedges. Unfortunately, the pressures are usually overestimated by an order of magnitude by this approach. A variety of reasons have been offered for this difficulty, but one that has been ignored is that this approach fails to recognize the true geometry of reentry. Figure 5b shows a plan of the ship at an instant during reentry. The dotted line represents the intersection of undisturbed free surface with the hull. This line is roughly the ridge line of instantaneous

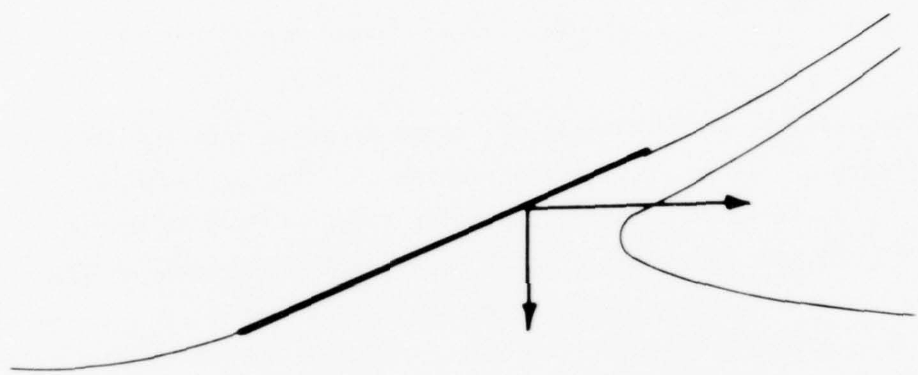


Figure 3 - Unsteady Planing

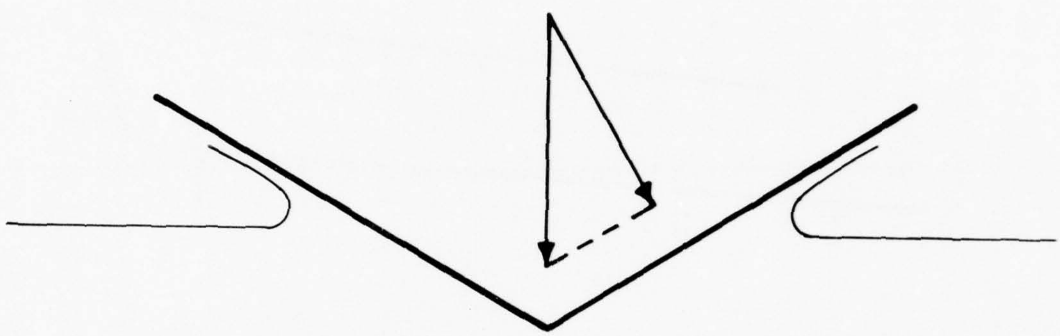


Figure 4a - Dropping Wedge

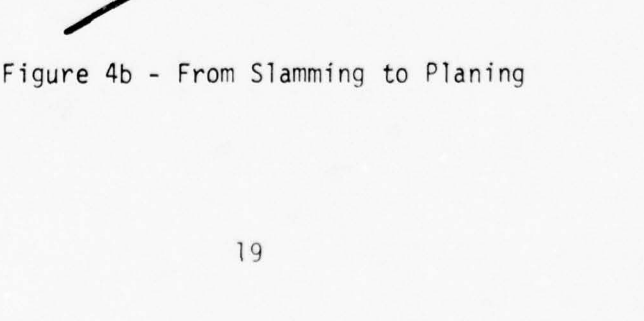


Figure 4b - From Slamming to Planing

Figure 5a - A Slam

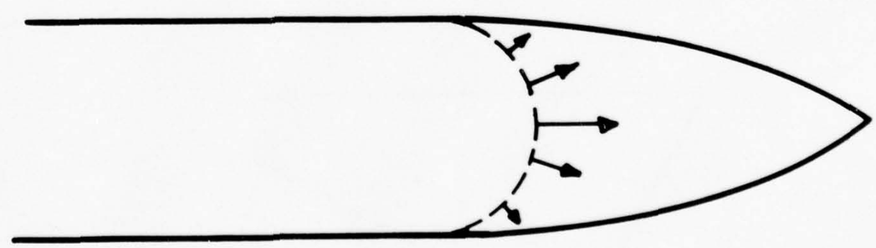


Figure 5b - Plan View of a Slam

peak pressure. It is moving forward and outward with time, and in the region where the pressure is near its absolute peak, it is obviously moving forward very fast. A two dimensional analysis which considers the local flow normal to this line might have a chance of success. But recognizing that the angle between the hull and the surface is a maximum along such a normal, it is evident that the much smaller angle in the transverse plane, as would be the consequence of the "strip theory", would lead to an overestimate. (The peak pressure is roughly inversely proportional to the square of this angle).

Now take the planing surface and give it a slight dihedral, thus converting it into a prismatic wedge, Figure 6. No real theory is available for this three dimensional case, but there is a mass of steady state data. Again, it would be an important challenge for numerical hydrodynamics. One aspect of the problem might actually be simpler than for the two dimensional case -- the spray is thrown to the side and thus can be considered to leave the flow field.

Next, let us raise the fore and aft angle between this wedge and the water to near 90 degrees, rather than near zero, and let the wedge extend deep into the water, Figure 7. We now have a model for the very bow of a ship, a region of great interest and importance which has received virtually no theoretical attention and very little experimental treatment. If we introduce a transverse velocity, the problem is of even greater practical importance, because a ship does not really move steadily forward on its axis -- the bow wanders from side to side, either because of steering errors or transverse waves. The local flow is important because a low pressure region on the side of the bow provides a passage for bubbles to be entrained, to be carried down to where they can interfere with sonar operation. In an extreme case, steady state or intermittent ventilation can occur, with potentially serious effects. Introducing a bow radius is all that is needed to achieve a high degree of realism.

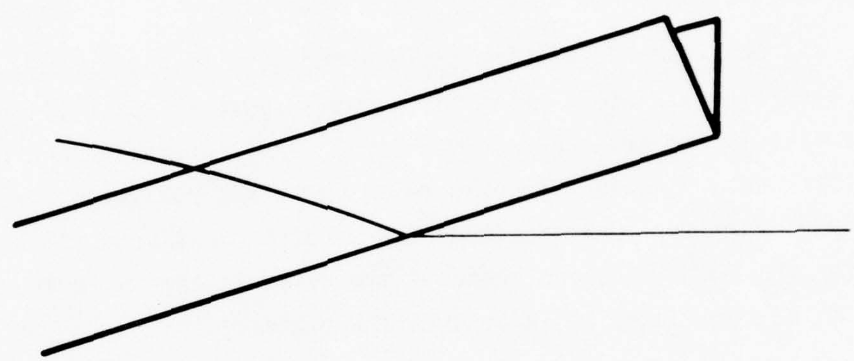


Figure 6 - A Planing Prism

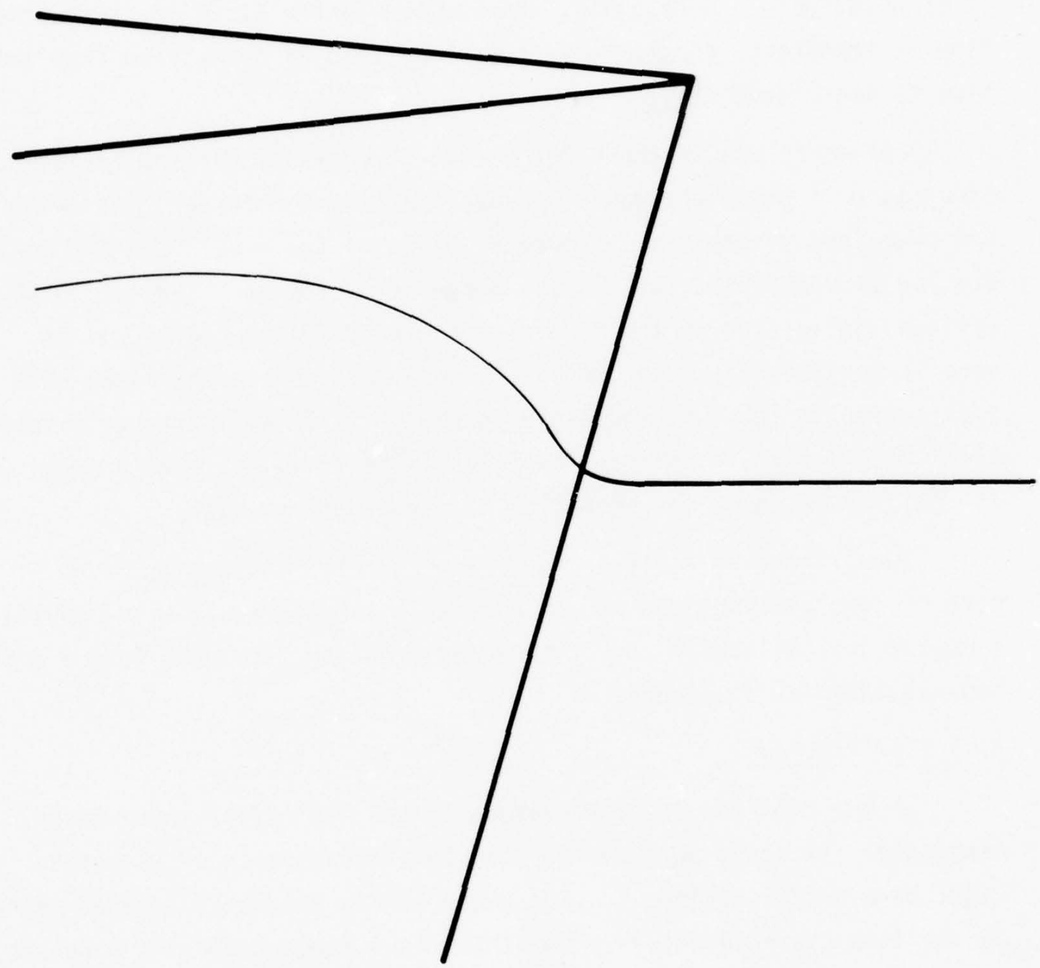


Figure 7 - Bow Entrance

From this problem it is only a step to the surface piercing strut, with its very extreme problem of ventilation, Figure 8. This subject is of such great importance to the hydrofoil craft, and possibly other advanced craft, that it is receiving extensive experimental treatment. A valid theoretical or computational approach could be of great help. Here again, the passage to the strut in transverse flow is important, particularly the prediction of transition from wetted flow to ventilated flow.

Let us return to the original two dimensional planing surface and carry it well below the water surface, letting the forward jet swing up, over, and to the rear, Figure 9. This is the model of a supercavitating or ventilated foil in the presence of the free surface, with obvious application to the very high speed hydrofoil craft. If we move it sufficiently deep, we have a pure supercavitating flow, with application to the supercavitating propeller. If we introduce a vertical oscillation, either of the foil or the fluid, we have a model of the foil in waves or the propeller in a non-uniform wake.

Thus, there is a hierarchy of free surface and cavity flows, more or less related, and it would be most surprising if a successful computational attack on any of them could not be developed into a more general approach to others.

Real Flow Problems

In the previous section, viscosity and its effects were never mentioned. It would be expected that the introduction of viscosity might be a useful refinement, but would hardly result in a major change in the flow characteristics. But there is a large number of problems in which viscosity is the dominant factor, or in which viscosity triggers some effect which completely changes the nature of the flow. A collection of these will be presented in this section. Here, again,

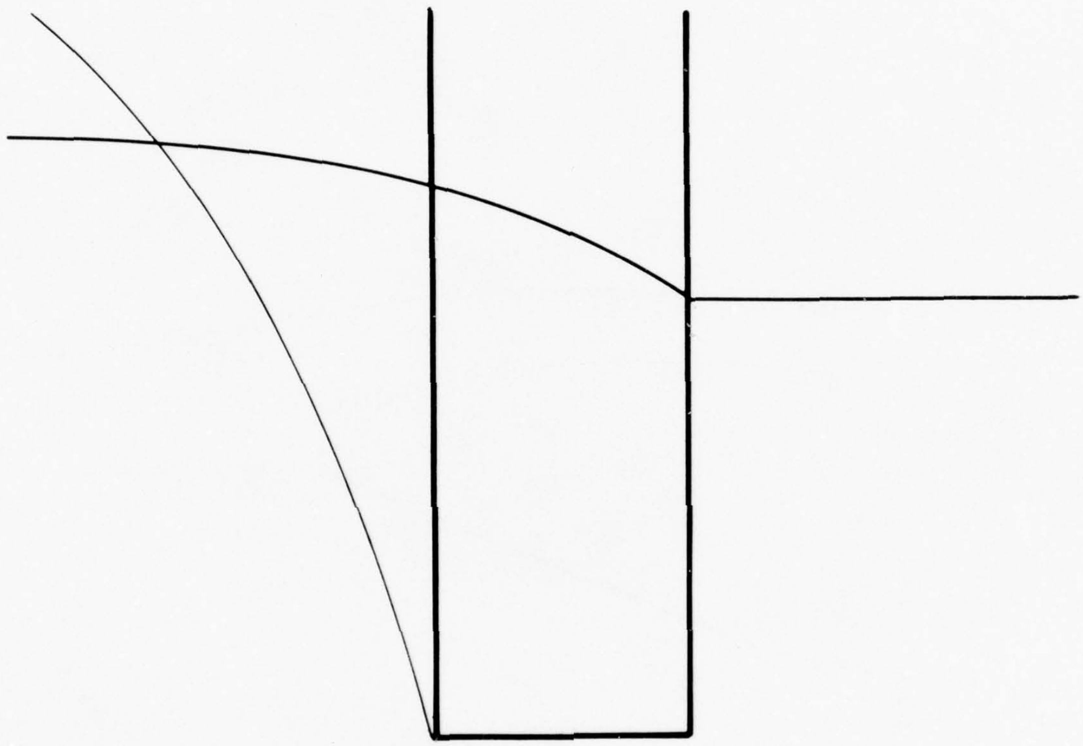


Figure 8 - Surface Piercing Strut

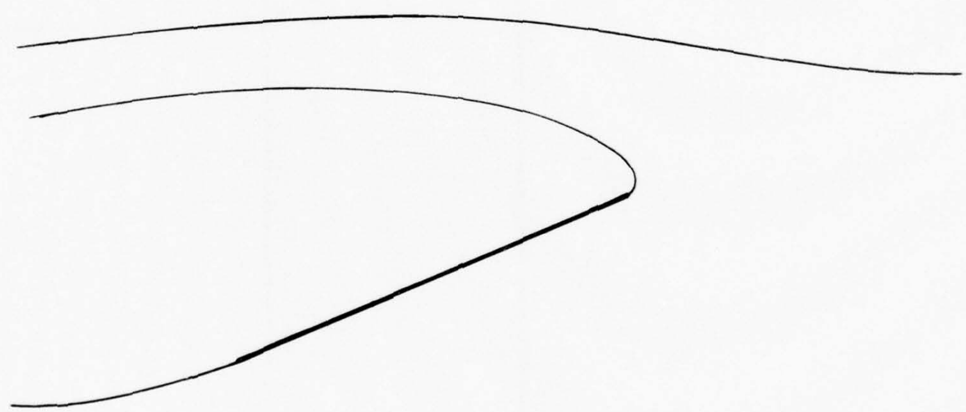


Figure 9 - Foil with Cavity

it might be hoped that certain general approaches could be developed that would permit a progressive attack.

It is appropriate to start off with the problem of transition. This has already been introduced in the discussion of "global" problems, but it would appear to make better physical sense to consider it a local problem. What is needed is a method of analysis, or better yet, a set of criteria which permits the pinpointing of transition line -- on the flat plate, on the body of revolution, on an arbitrary hull. How sensitive is this line to free stream turbulence, to pressure gradient, to noise, to vibration, to transverse curvature, to longitudinal curvature, to rate of change of radius (for a body of revolution), to a heated wall, to surface roughness, and to any other possible triggers? Many of these questions have been addressed, and traditional boundary layer theory seems to offer considerable hope for success. However, the total problem must still be considered unsolved.

Every proturberance, sudden change in shape, or opening on a hull or appendage is a potential vortex generator. We have vortex sheets, Karman Streets, tip vortices, hub vortices, and probably a number of types that have not yet been described. All of these are potential sources of trouble.

Vortices aligned with the flow imply low pressure cores which have strong tendency to cavitate. Also, the high energy of these vortices may have a drastic effect upon any downstream appendages upon which they impinge. The PCH hydrofoil craft, which has a propeller forward of a strut and foil, had a serious erosion problem where the tip vortices impinged on the foil. The effect of the tip vortex from a submarine sail, as it sweeps across the stern appendages, is suspected to be a factor in snap roll. Vortex cores from active stabilizing fins look like ropes of cavitation which extend far downstream. The phenomenon of such vortices is well understood, and in some cases

we are able to design to alleviate the effects. Nevertheless, a general computational tool could be of significant value.

The vortex trail transverse to the flow, or Karman Street, is of even greater importance because it can so easily excite vibration. Propeller singing is a consequence of a coupling between the vortex shedding frequency in the propeller blade wake and some structural response of the blade itself. A similar source of trouble is the Helmholtz phenomenon which is a potential problem at any opening in the hull. Eddies from the upstream edge can interact with some mode of structural response within the opening itself. The current practice is to seek a simple "fix" for any such problem, based upon the most primitive understanding. Again, there is an opportunity for more refined computational procedures to permit more effective design.

Propellers are usually designed by means of potential flow theory, using mean wake characteristics. Actually, of course, they generally operate in a widly fluctuating environment, and any blade is usually in an off design condition at any instant. The consequence is sometimes heavy vibration and propeller blade fatigue. A means of predicting blade loading would be of great value. An even more challenging case is that of the propeller thrown into reverse for a "crash stop", where the loadings are known only to be large. Failures have occurred.

CONCLUSION

This review of the unsolved problems of ship performance prediction has been oriented toward the opportunities for numerical hydrodynamics. It is recognized that most of these problems are at present beyond the capability of these new techniques. In particular, most applications involve Reynolds' Numbers far above the range so far treated.

It may be predicted that the ultimate solutions will be the result of some sort of mixed strategy, in which the more conventional

approaches are used to scope a problem and to define the local conditions which are then treated in detail by the new procedures.

An example might be one of the standard problems of seakeeping, the specification of the magnitude of slamming loads for structural design. The traditional linear methods may be used to isolate in time and space a set of situations in which severe slams would be anticipated. These methods can even provide gross estimates of the magnitude of the slam. But the prediction of the detailed time history is far beyond them. When these new numerical procedures are sufficiently advanced, we should be able to examine the small sample population in depth, and arrive at meaningful design criteria.

At NSRDC, for the past year, we have been making the opening moves in this new game. Our progress will be reviewed in a later presentation at this workshop. These are learing steps, carefully selected to open new paths rather than to achieve immediate breakthroughs. But we are pleased with our progress, and we look forward to the challenges of the future.

REFERENCES

1. Wu, J. and L. Landweber, "Variation of Viscous Drag with Froude Number," Proceedings of the 10th International Towing Tank Conference, Teddington, England, September 1963.

2. St. Denis, M. and W.J. Pierson, "On The Motions of Ships in Confused Seas," Transactions of the Society of Naval Architects and Marine Engineers, Volume 61, 1953.

3. Cummins, W.E., "Pathologies of the Transfer Functions," Proceedings of the Seakeeping Symposium Commemorating the 20th Anniversary of the St. Denis-Pierson Paper.

4. Sedov, L.I., "Two-Dimensional Problems in Hydrodynamics and Aerodynamics," Interscience Publishers, 1965.

DISCUSSION

G. F. Carrier

It seems to me that in all solid hydrodymanics there are two basic kinds of problem that one faces: (1) Those in which we understand very well the mechanics involved, the comparative importance of individual mechanisms and the physics which underlies them, but the overall problem is so intricate that terrible computational difficulties result, and (2) those in which our understanding of the mechanisms and the underlying physics is so inadequate that we cannot even formulate the problem in an intelligent manner. I must admit that I am not current on all that you have discussed, but I have the distinct impression that most of those problems fall into my second category. Is this true, or am I just behind the times?

W. E. Cummins

I believe we understand rather well the mechanicms underlying those problems I have discussed.

G. F. Carrier

Do you understand cavitation well enough?

W. E. Cummins

Fully developed cavitation, I believe, is well enough understood to be able to describe and treat adequately. The inception of cavitation, however, is a different matter. Our understanding of this process is very poor. You can miss a cavitation speed on a propeller by a factor of two. We don't know why we are off.

G. F. Carrier

In other words, cavitation is understood under idealized conditions, but what is needed for your computational models is a full understanding

Preceding Page Blank -

under realistic operational conditions. This leads me to my point, that of wondering if improved analytical or numerical methods can really be fully applied as long as our understanding of the underlying physics is incomplete.

W. E. Cummins

In some cases we do need better physics.

G. Moretti

I would like to emphasize what you said, Professor Carrier. I think we should try to understand what we are doing, when we make numerical experiments. In this way one can frequently learn a great deal of physics. For instance, in making numerical experiments a pattern will occasionally show up that had not been predicted at the level of our current understanding. So, you try to remodel the experiment in the next stage so that the numbers describe the new pattern. This is a very useful approach to understanding the underlying principles as long as we do not delude ourselves by placing excessive confidence in every detail of our numerical description.

J. Buck

I am curious to know how you arrived at the selection of priorities in your problem areas. You identified your global-type problems which seem to emphasize ship motion, but are directed towards conventional ship designs. On the other hand, your local problems seem to emphasize hydrofoil applications. I wonder if you could explain how you arrived at this set of problems in the light of present and future Navy requirements.

W. E. Cummins

I listed the global problems, and identified them with conventional ships, because they are still with us. We haven't made a great deal of progress towards an ultimate solution to any of them. And, I might add, anything we can get from numerical hydrodynamics that will help us to

better understand these problems will be most useful. However, since, all things considered, we do know how to design conventional ships rather well, the greatest needs for the future will lie in the area of high-performance craft. In this area we have poor to nonexistent facilities, inadequate theory, extremely difficult geometries, and very high needs because we are faced with having to design with precision in an area where we have only the crudest sorts of tools. I believe that it is in this area that the greatest future of numerical hydrodynamics lies, and this is the reason that I concentrated in my paper on these problems. If numerical hydrodynamics is going to require massive funding I don't see that funding being made available to an area we already do fairly well -- namely, the design of conventional ships.

S. I. Cheng

On the question of physical models versus the analytical approach, I feel that we should combine them into a complementary attack on the outstanding problems. Perhaps, this way, we can eliminate such problems as nonlinearity.

G. F. Carrier

Judging from some of these comments the point I was trying to make earlier may have been misunderstood. I was not trying to suggest that numerical methods are going to save the day where analytical methods fail. As far as I am concerned there are problems which are well enough formulated so that now all we need to do is solve them and I don't care whether we do that numerically or analytically. But there are other problems where we don't have a well-set mathematical formulation, and no solution method can resolve that difficulty. I suggest that we are in this latter category. In other words, we need a better understanding of the underlying physics, or at least we need to be able to construct a gross empirical description of the mechanisms involved, before we can even formulate the problem. Then, whether we use analytical or numerical methods merely depends on the complexity of the problem. There is no issue between the two that I can see.

G. Birkhoff

I will agree that some, though not all, of the problems fall into Professor Carrier's last category. It would be very interesting to try to determine where uncertainties exist in things like boundary layer transition (to turbulence) and flow separation. The determining factors are nortoriously more difficult to define for such problems. On the other hand, in problems such as ship slamming I believe we have a sound basis for mathematical modeling.

W. E. Cummins

There are some arguments on that, by the way, but I agree with you. Many people, for instance, think that the air cushion between hull and water is a very important factor. However, I believe that the problem is rather well formulated.

M. Tulin

I was struck by the fact that in your catalog of free-surface problems you didn't mention the problem of predicting the wave resistance of a ship. Is there a reason for leaving that out?

W. E. Cummins

Yes. First of all, I think wave resistance is a very interesting problem for the mathematician. I don't consider it to be a terribly interesting problem for the designer. My discussion, of course, was directed towards the problems of practical design. For design purposes we do need some information about what happens right at the bow. In general, however, we do not need information beyond that we already have about the wave train. In separating wave resistance from total resistance, we do pretty well. We don't pretend that our techniques are precise, or scientifically valid. But, the technique is adequate for our purposes. We can predict within five per cent and they don't build ships with greater precision than that. What we do now is measure total resistance and estimate the frictional resistance as well as we can using drag

data from flat plates, for example. We take the latter quantity out and treat the residual resistance as if it were wave resistance (i.e., it would scale by the Froude Law). We know that is wrong. We also know that if we are careful in those areas where we have experience we get answers we can use for design. So, while I consider the problem of predicting wave resistance a fascinating one, I don't think it is going to be one to stimulate major support for numerical hydrodynamics.

I might add that none of the linear theories have been adequate to define more than the general characteristics of wave resistance, nor, although Hogben came close to it on an air-cushion vehicle, has anyone ever been able to measure wave resistance. It is a very difficult problem.

M. Tulin

You feel, then, that in the absence of an adequate body of theory upon which to base prediction the present experimental methods are sufficient to your design requirements?

W. E. Cummins

That is correct.

G. Birkhoff

I am not a proponent of the massive funding approach to problem solving. Warren Weaver, a man of great experience as most of you know, used to say that well-organized engineering groups usually dealt with about five or six uncertainties that had been reduced to about the same order of magnitude. Weaver felt that to concentrate on reducing one uncertainly would make little practical difference since the other four or five would still remain.

W. E. Cummins

Yes, it is similar to using the noise from a noisy source. Incidentaily, I would like to comment on your statement about massive funding. It might be helpful to put that in perspective. The total funds available

today are not very large. Nor, in the areas of basic and applied research, do they need to be massive in order to be effective when compared against what is available. I think a better term for what is needed would be sustained funding.

M. Tulin

Your catalog is both useful and provocative. To what problems on your list would you assign the highest priority?

W. E. Cummins

Those that are the easiest to deal with. All of these problems interact in such a way that it is important to make early progress. It would be easy, and perhaps defendable, to assign the highest priority to one of the more difficult problems. However, I think it is more important that we learn to walk before we try to run and broadjump. Some of the problems further down my list will require very long broadjumps.

M. Tulin

When you say further down the list do you mean you listed them in the order of increasing complexity or importance?

W. E. Cummins

Complexity. For example, the strut problem is the most important one on the list.

M. Tulin

I was impressed, though, that the first problems that you discussed all involved separation. This, of course, brings us to Professor Carrier's point. In talking about not knowing physics in relation to ship problems we essentially mean problems involving turbulent boundary layer and separation, vortex shedding, and the prediction of the length of a finite cavity. So, those first problems will certainly be very difficult.

W. E. Cummins

Agreed. I don't think any of the global problems will be easily solved.

G. F. Carrier

I would add that it is not just the fundamental physics of separation or transition that is not understood; we don't even have good empirical rules that can be used to fill in one of your global problems. I don't insist that we understand meticulously, but we must have something which, if used successfully, has meaning when the answer comes out, even if it is highly empirical. We don't even have this in many cases.

H. Elrod

The problem is complicated by the fact that while we may recognize the factors involved we don't always know what weight to give them. For instance, the problem of spray resistance of full-formed ships was ignored for a long time despite the fact that there is plenty of spray in front of ships. Some years ago one would be likely to leave this factor out of a numerical calculation on the resistance of ships. As another example, Dr. Cummins mentioned the problem of slamming in which the captured air underneath the hull may be important to structural response. We have a long history of dealing with problems in which the physics wasn't put straight at the outset.

W. E. Cummins

Yes, and a good example of this is cavitation inception, or even a bit further along when the cavity is better developed. We always assumed that the reasons we weren't able to do well with the towing tank was because we were testing at the wrong pressure, and that a variable pressure towing tank would solve the problem. Well, there are two of these now and we have done a great deal of work at the Lockheed facility in California. We quickly found that we got no better results there than we had in the

towing tank. So, changing the atmospheric pressure was not the factor we thought it to be -- we simply missed on the physics. Now, our best guess is that Reynolds number is the single most important factor. So, somehow we have to try a run at the highest Reynolds number we can get away with in light of the other boundaries on the problem.

T. Sarpkaya

First, I would like to compliment Dr. Cummins on an excellent summary. His compartmentation of unsolved problems into global and local categories is an effective technique. Since both basic and applied fluid dynamicists know which problems we understand to varying degrees and those we don't, it might also be useful to separate the list of problems into those where we have a clear understanding of the physical phenomena and those we don't.

W. E. Cummins

As a final comment I would like to repeat that I have been offering opportunities, not promises.

THE NAVY'S HYDRODYNAMICS PROBLEMS:
A DESIGNER'S VIEW

Robert S. Johnson and Peter A. Gale
Naval Ship Engineering Center
Hyattsville, Maryland

ABSTRACT

This paper describes the most important unsolved hydrodynamic problems for conventional monohulls and high performance ships, from the naval ship designer's viewpoint.

INTRODUCTION

The objective of this paper is to list and describe some of the most important current problems in hydrodynamics facing the ship designer. The problems mentioned are those for which the designer most urgently needs solutions. The authors, being ship designers, certainly have a somewhat different yardstick for measuring importance than would shipbuilders or research scientists. This should be borne in mind when reading the paper. Scientists might know that some of the problems listed herein have already been solved. This would not be surprising. Quite often, the designer is either ignorant of an existing problem solution or cannot utilize the solution for some practical reason.

The paper is divided into two main sections. The first section discusses problems associated with conventional monohull displacement ships while the second treats the hydrodynamic problems of several high performance ship types of current interest. Ship types not addressed in the paper include submarines and conventional multihulls,

including catamarans. Low waterplane area twin hull ship hydrodynamic problems are discussed in the high performance ship section. Topics not addressed in the paper include acoustics and infrared (IR) signature problems. It should be noted that the problems which are mentioned are not listed in any order of importance.

Ship designs can be classified as either "well understood" or "poorly understood." By "well understood" is meant designs which do not differ notably from previous successful ones from the standpoints of configuration, technical complexity, etc. "Poorly understood" designs are those which reflect novel configurations and present technical problems previously unsolved. They generally represent significant advances in the state-of-the-art. Most conventional monohull displacement ship designs are "well understood" while most high performance ship designs are "poorly understood." The design process and the nature of the hydrodynamic problems faced during design differ significantly for these two classes of designs. The problem descriptions in the paper make this evident. Today, five phases of design are identified for "well understood" conventional ship designs:

Feasibility Studies - emphasizing requirements trade-offs

Conceptual Design - emphasizing absolute sizing of the ship

Preliminary Design - emphasizing ship system optimization and definition

Contract Design - emphasizing development of the procurement specification

Detail Design - emphasizing development of working drawings

In Appendix A, each of these design phases is described for "well understood" conventional surface ship designs including objectives, nature of work performed, and their hydrodynamic interfaces. The early phases of design, Feasibility Studies and Conceptual Design, will be quite different from the Appendix A descriptions for an advanced concept or a "poorly understood" conventional design. Where knowledge of the concept

is deficient, these two phases will emphasize risk identification and design tool development in order to lay the basis for further ship design development. "Proof of principle" experiments may have to be conducted in the early design phases.

The nature of the ship designer's calculations changes as a design is developed. In the early design phases, system performance must be estimated on the basis of incomplete or gross data. In the later design phases, detailed calculations are done for the purpose of precisely specifying hardware; e.g., a flush waterjet inlet's shape. Thus prediction or analysis techniques which are well suited to one phase of design may not be useful in another phase. Frequently, a method of solution for a complex problem exists which is invaluable for later phase design calculations but is too cumbersome and costly for use in performing early phase trade-off studies. Or it may not be possible to use the method directly because too much design definition is required as input to it.

In general, economics dictates that early phase ship design may not rely on experimental methods. This is because such methods are both expensive and time consuming. This statement applies to both conventional and unconventional designs. Analytical methods, computer assisted or manual, must be used to sort out the most promising configurations in the context of the overall ship design. Similarly, reliance must be placed on analytical methods to screen out undesirable candidates. Analytical methods are used to study the sensitivity of the baseline concept to various parameters. Once a concept has been established as promising "on paper", then a model can be built and tested. A reasonably thorough (but not complete) model test program for a new high-performance ship concept will cost more than a quarter of a million dollars. Even the cost of the straight-forward model tests for conventional monohulls exceeds $100,000. Despite the expense of model tests, they cannot be dismissed now or in the foreseeable future, because analytical methods

fall short of the engineer's needs. However, at current prices they must be applied selectively to achieve specific program objectives.

Before turning to the main text it should be noted that ship designers are, to some extent, becoming "criteria limited." That is, in some cases we can now do more sophisticated calculations, e.g., ship motions, than is merited by our ability to judge whether the answers are acceptable. For example, is accepting "one slam per minute" the proper criterion; is it even the proper way to state the criterion; and, what defines a slam?

CONVENTIONAL MONOHULL DISPLACEMENT SHIPS

Calm Water Resistance and Propulsion

General Approach

Model testing is not generally initiated until late in the Preliminary Design phase, if then. Prior to the availability of model test results, calm water speed/power characteristics of alternative hull forms under consideration are predicted by empirical means.

Generally, bare hull resistance predictions are based upon Taylor's Standard Series and are modified to account for differences from the Taylor hull form. Figure 1 outlines the approach used. Other standard series data are used when available and when the Taylor parent hull is not appropriate, such as for high displacement-length ratio hulls, high maximum section coefficient hulls, etc. In some cases, estimates are based on specific parent hull forms or regression analyses rather than on standard series.

In the early design stages, appendage drag estimates are based on model test results for similar ships tested with and without appendages. The total estimated appendage drag is expressed either as a percentage of the bare hull drag or as a drag coefficient in empirical relationships derived from available data. When approximate appendage configurations have been developed, final estimates of total appendage drag (prior

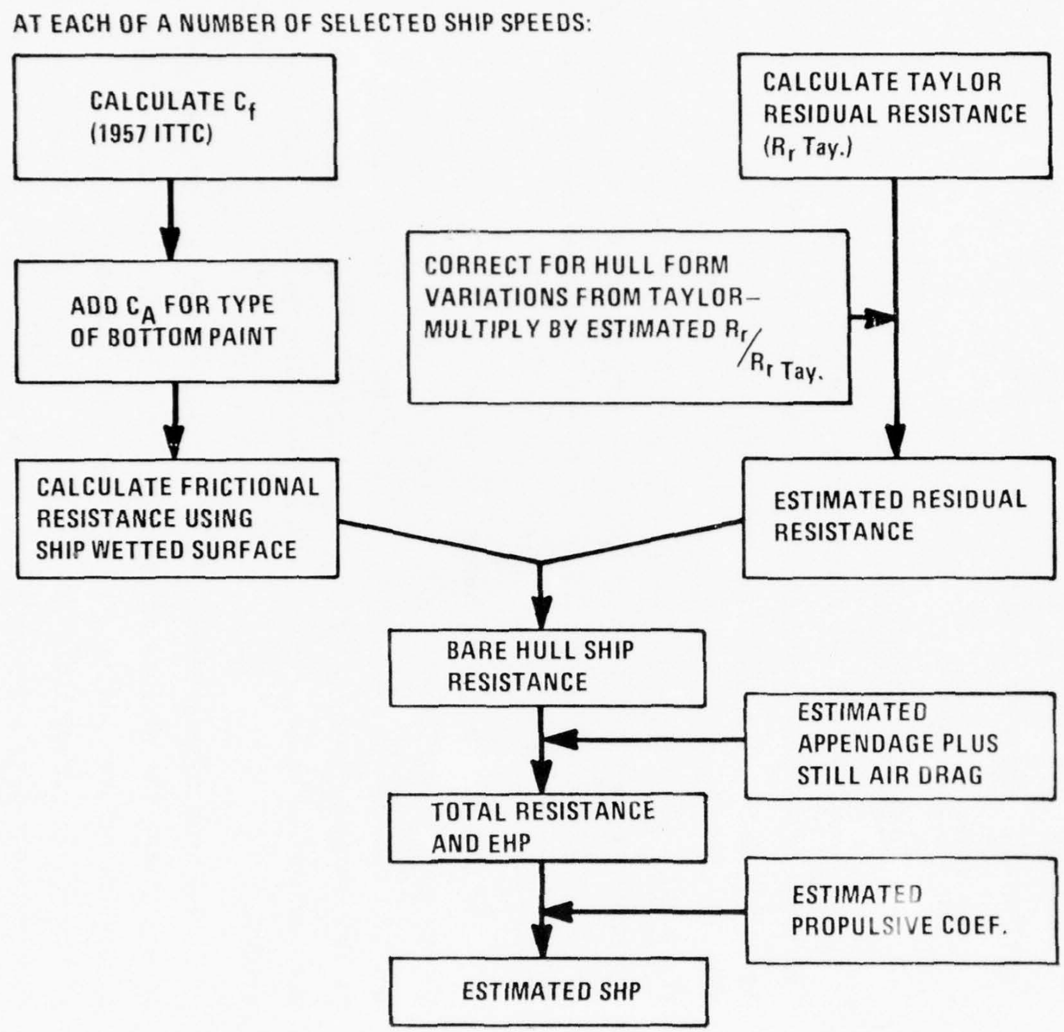


Figure 1. General Approach to Calm Water Speed/Power Estimates

to model testing) are based on the predicted drags of the individual hull appendages.

Initially, overall propulsive coefficients are estimated. Later, but prior to model testing, the open water propeller efficiency is calculated from propeller series data and the other individual components of the propulsive coefficient (η_{rr}, "w", and "t") estimated from empirical relationships based on data for similar ships.

Specific Problems

1. Bare Hull Resistance Prediction

Improved methods of predicting bare hull resistance are needed. The usual procedure, based on Froude's method and using Taylor's Standard Series as outlined earlier, is generally satisfactory for normal hull forms even though the technique is known to be scientifically incorrect. The method fails when predictions for hull forms unlike those models tested previously are desired. This is because correction factors derived from model tests of similar hull types are essential in using the method.

In today's cost conscious environment, the merits of hull forms incorporating developable surfaces and multichines are studied in the course of virtually every major new ship design. At present it is necessary to conduct model tests even to approximately assess the drag of such variants; valid theoretical or empirical prediction techniques are needed.

The traditional resistance prediction method is also poorly suited to evaluating the effects of changes in the longitudinal distributions of waterplane area and immersed volume. This is another aspect which is studied in connection with most major ship designs. These longitudinal distributions have a strong effect on hull shape (bulb, entrance angle and transom size) and, thus, on internal arrangements and stability as well as seakeeping characteristics and calm water resistance. The effects

of such hull form variations on ship motions in head seas can be predicted quite well at present but rational trade-off studies are hindered by the lack of valid methods of evaluating the effects on calm water resistance. Time and cost constraints generally do not permit model testing the necessary number of alternatives. For certain specific hull types, regression techniques can be used to make predictions within specified bounds on each of the key variables. However, these methods can easily be ±5% in error and this is unsatisfactory.

Another aspect of this situation is worthy of mention. Hydrodynamicists are rapidly developing methods of calculating "optimum" (minimum drag) hull forms for given boundary conditions as well as the "optimum" bulb and/or transom shapes for minimizing the drag of given hull forms. Unfortunately, at present their ability to predict optimum shapes is far better than their ability to predict the drags of those shapes. This inability to predict resistance has been a major factor tending to restrict the adoption of such optimum hull forms. Ship Design Managers are naturally reluctant to commit substantial funds to model testing "radical hull forms" without first seeing convincing performance predictions made by their advocates.

2. Appendage Drag Prediction

Reasonably satisfactory methods exist for predicting model appendage drag before model tests are conducted. All such methods are either based on the analysis of model tests with and without appendages or are correlated with such tests. The major problem associated with appendage drag is the unknown relationship between the model test's results and the actual ship performance,i.e., the scale effect problem. It is well known that substantial appendage drag scale effects exist but the naval architect does not possess generally accepted methods of correcting for such effects in his full scale powering predictions. Figure 2,

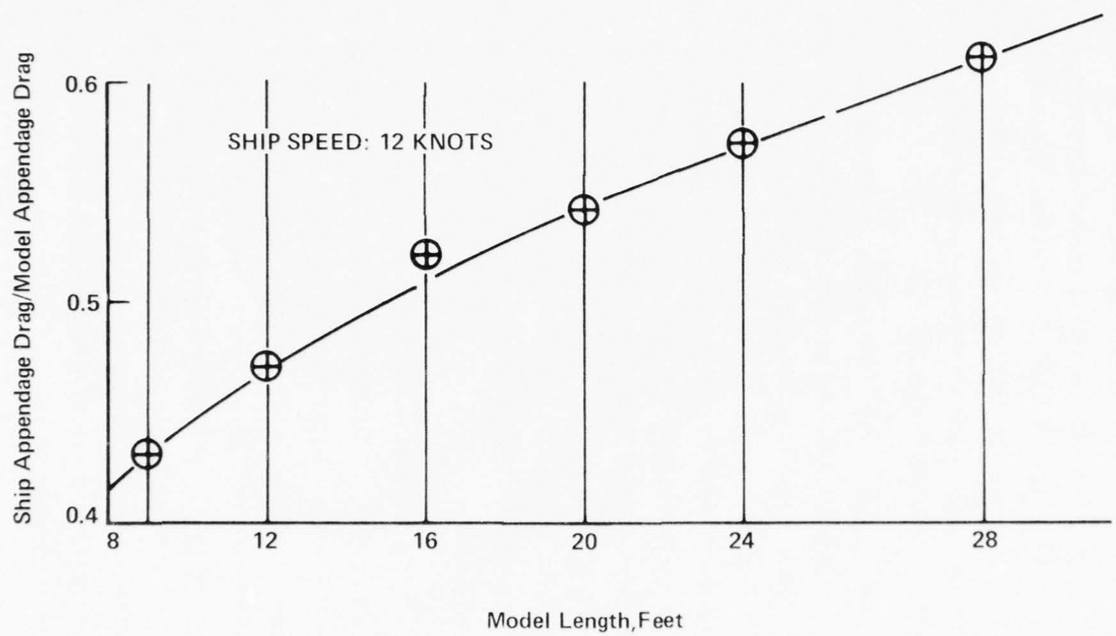


Figure 2. Ship/Model Resistance Increment For Twin Shaft Bracket as a Function of Model Length

based on results published in Reference 1 is included to show the scale effects associated with the drag of a particular type of appendage. The figure indicates that, for the usual model lengths of 12 to 20 feet, the shaft bracket drag on the subject ship is roughly one-half of that on the model.

3. Propulsive Coefficient Prediction

For normal designs, speed/power estimates in the earliest design stages use propulsive coefficients based on model test data from previous similar ship designs. As a design is developed, more refined estimates of propulsive coefficient are made by assessing the individual components of the overall coefficient. Open water propeller efficiency is estimated from propeller model series data (Troost or other) or by actual propeller design calculations. In either case, the accuracy is quite satisfactory. There is a developing need for systematic performance data on high pitch to diameter ratio and large hub diameter to propeller diameter ratio propellers. This latter requirement stems from the increasing use of controllable pitch propellers.

There are several major problems associated with predicting a propulsive coefficient, all related to components of the overall coefficient other than open water propeller efficiency. Prior to model testing, the values of relative rotative efficiency (ηr), thrust deduction factor (t), and wake fraction (w), are estimated based on model test results for similar ships. Problems encountered include:

a. Considerable variations in collected model test data with no apparent explanations.

b. Lack of knowledge regarding major influencing characteristics makes interpretation of available model test data difficult even when making estimates within the bounds of existing data.

c. Virtual impossibility of making estimates for new designs whose propeller, appendage or hull form characteristics lie outside the bounds of existing data.

Problem b in the list above is by far the most important. If it were solved, problems a and c might well disappear. A few words to clarify problem b might be in order. The designer needs to know: (1) what hull form, appendage configuration and propeller characteristics have significant effects on "w", "t", and ηrr and (2), the quantitative influences of each of the significant characteristics on "w", "t", and ηrr. For example, for a typical twin screw destroyer hull, a model of which has been tested, which of the following characteristics strongly influence "w": C_x, C_p, B/T, L/B, displacement/length ratio, transom width/beam ratio, propeller diameter, propeller tip clearance, shaft rake, sonar dome size and shape, longitudinal separation between propeller disc and aftermost propeller shaft struts, and propeller shaft diameter? Are there important characteristics which are not listed? For each one of the significant characteristics, how will specific numerical changes in its value affect the measured "w" value? If the answers to questions such as these were known, the three problems listed would quite likely no longer be major ones. The puzzling variations in collected model test data could probably be explained. New ship predictions could be made with confidence by correcting existing model test data for the effects of specific numerical changes in certain of the significant characteristics. It is even likely that valid predictions could be made for hull and appendage configurations beyond the bounds of existing data.

Once a new ship design has progressed to the point where model tests are undertaken, another problem arises too often. This is the situation where ηrr, "t" or "w" varies considerably from one test to another for no apparent reason. For example, a propeller might be designed based upon values for these factors derived from a self-propelled

model test with a stock propeller and then, when the model test is repeated with the design propeller, large variations in "w" and "t" values are observed. When the principal characteristics of the design wheel are very close to those of the stock wheel (diameter, pitch, number of blades, blade area ratio, etc.) and the model hull and appendage configuration has not been altered, such variations are perplexing. Tables 1 and 2 present data on propulsive coefficient components derived from model tests on two recent naval ship designs. Table 1 shows clearly that substantial changes in propulsive coefficient can result from relatively small changes in the values of "w", "t", and ηrr (note that only about half of the total propulsive coefficient change is due to the increase in ηp). Table 2 is notable in that:

a. It shows a large, unexpected increase in wake resulting from minor appendage changes and the use of the final design propeller versus a stock propeller.

b. It shows the good repeatability of a retest conducted some four months after the original test.

The last problem which will be mentioned is scale effect. As with appendage drag, it is well known that substantial scale effects exist between model and full scale for several components of the propulsive coefficient, especially "w" and "t". As with appendage drag, these scale effects are generally ignored due to a lack of knowledge on how to quantitatively correct them in specific situations. Figure 3 presents some data regarding scale effects on propulsive coefficient components. The data on thrust deduction factor are taken from Reference 2, while Reference 3 was the source for the data shown regarding wake fraction. Although the Victory Ship 1/6 scale and the ALBACORE full scale data are not exact, clearly it is apparent that measured "w" and "t" are dependent upon model scale. The data point for the ALBACORE thrust deduction factor is of interest in that it shows the only known full scale value and the

1-t	0.90	0.93	0.92
$1-w_T$	0.97	0.98	0.96
η_{rr}	0.98	1.00	0.99
η_p	0.66	0.68	0.69
P.C.	0.59	0.64	0.66
Remarks	Stock Prop.	A Design Propeller– Minor hull, propeller diameter and appendage changes	Final Design Propeller – Minor appendage changes

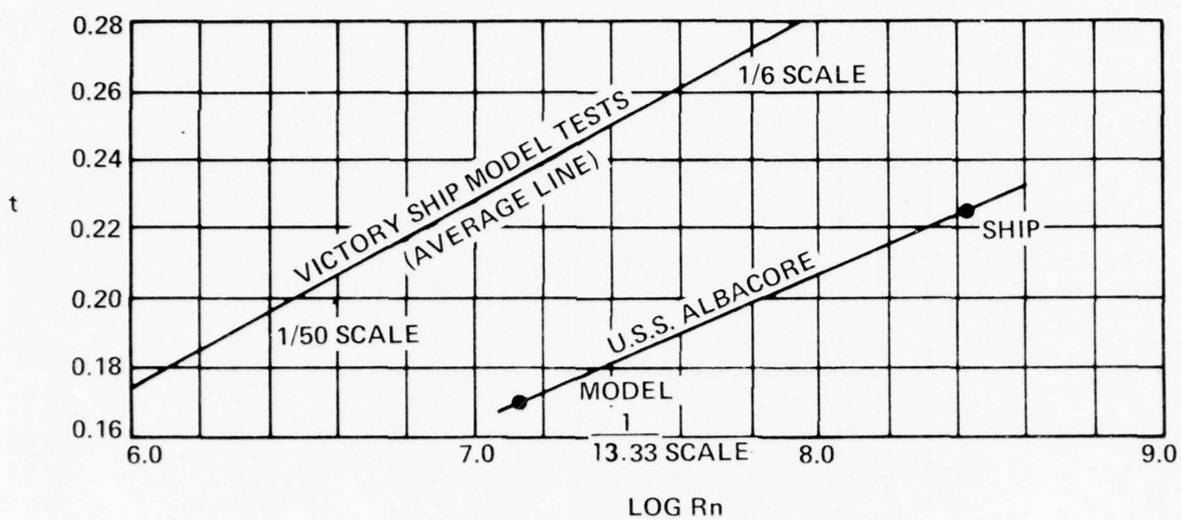
Note: All numerical values rounded off to the nearest hundredth.

Table 1. Propulsive Coefficient Component Comparison No. 1 (at a specific speed)

1-t	0.91	0.90	0.89
$1\text{-}w_T$	0.99	0.99	0.95
η_{rr}	0.96	0.97	0.96
η_p	0.69	0.69	0.70
P.C.	0.62	0.61	0.63
Remarks	Stock Propeller	Retest—(configuration identical)	Final Design Propeller--Minor appendage changes

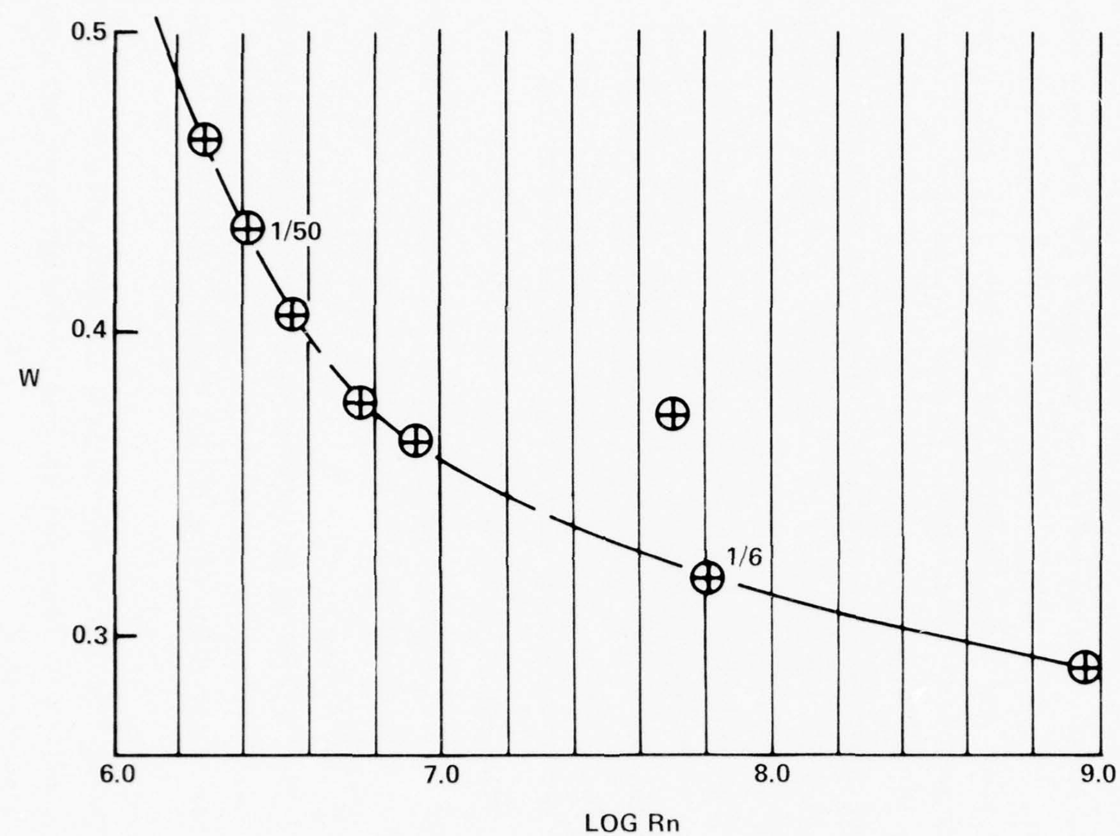
Note: All numerical values rounded off to the nearest hundredth.

Table 2. Propulsive Coefficient Component Comparison No. 2 (at a specific speed)



Variation of thrust-deduction coefficient with scale

Figure 3a. Wake and Trust Deduction Scale Effects
(at a specific speed)



Variation of nominal wake fraction with scale

Figure 3b. Wake and Thrust Deduction Scale Effects
(at a specific speed)

slope of the trend line confirms the trend demonstrated by the Victory Ship geosim tests.

There are two principal reasons why the problems associated with predicting the propulsive coefficient are particularly important. One is the direct relationship between the propulsive coefficient and ship size, cost, and performance (effect on endurance SHP and hence required fuel weight, ship size, displacement, and operating costs; effect on achievable speed and hence installed SHP and ship size, displacement, and acquisition cost). Substantial changes in predicted propulsive coefficients in the later stages of design can result in very costly design changes and consequent delays or a ship which is either: (1) overdesigned and hence too expensive, or (2), fails to meet the required performance specifications. The other reason for the particular importance of the problems discussed above is the direct relationship between the propulsive coefficient components and the propeller design. Propeller design and the subsequent model test evaluations comprise a costly and time-consuming process. Last minute surprises which necessitate recycling of the design can be very costly in terms of both time and money.

In summary, there is a strong need for:

1. An understanding of the principal hull, appendage, and propeller configuration characteristics which influence "w", "t", and η_{rr} and their quantitative effects.

2. Better methods of estimating these three components of the propulsive coefficient prior to model testing both to improve the accuracy of early speed/power predictions and to identify spurious model test results.

3. Rational methods of correcting, for scale effects, values of propulsive coefficient components derived from model tests and used to predict ship performance.

4. Systematic performance data on high pitch to diameter ratio and large hub diameter to diameter ratio propellers.

Stability and Control

General Approach

The lateral plane stability and control aspects of new surface ship designs receive less attention during design than other aspects of hydrodynamic performance. Among the reasons for this are:

1. A lack of comprehensive, reasonably accurate, and easy to use theoretical or empirical prediction methods.

2. In general, a lack of specific, critical control requirements.

3. The relative ease with which deficiencies identified during model testing can be corrected by appendage alterations (rudder, skeg, etc.).

In the Conceptual Design Phase, the Top Level Requirements pertinent to a new ship's stability and control characteristics are developed. This is generally done based on past experience and without benefit of actual performance predictions for the design alternatives under consideration. The specific Top Level Requirements which are developed are generally restricted to: (1) a statement requiring the new design to be dynamically stable when steaming ahead, and (2) a maximum acceptable tactical diameter at full power and with full rudder. This latter requirement is generally based more on what is readily achievable than on specific operational requirements.

In the Preliminary Design Phase, a preliminary appendage configuration is developed and, at the end of the phase, preliminary performance predictions are made, usually without the benefit of model tests. In developing the preliminary appendage configuration, the rudder area and skeg length selected are primarily influenced by stability and control

considerations (see Figure 4). In most cases, the values selected are based on the geometries and associated performances of previous ship designs rather than on analysis. The same is true for the preliminary performance predictions made at the end of the Preliminary Design Phase.

During the Contract Design Phase, the appendage configuration is developed in greater detail and model tests are conducted. Typical tests in the stability and control area include dynamic stability, turning circles, and ahead and astern controllability tests. Deficiencies revealed by the model tests are corrected, when possible, by appendage modifications.

Specific Problems

1. Rudder Skeg Sizing

Analytical techniques to permit the rational sizing of rudder and skeg in the Preliminary Design phase are needed. Such techniques will:

a. Reduce model testing time and cost.

b. Permit the assessment of alternative appendage configurations on stability and control characteristics.

c. Permit early performance estimates and appendage design studies to be made for unusual hull and appendage configurations without resort to model tests. As has been mentioned, initial rudder and skeg sizing at present is largely based on past practice. The development of analytical evaluation techniques has been hindered by:

(i) Lack of financial support.

(ii) The ability to get by with "rule of thumb" methods.

(iii) Problem complexities: boundary layer, scale, and propeller effects, non-linearities, and the extreme complexity of the flow behind a self-propelled ship in a turn.

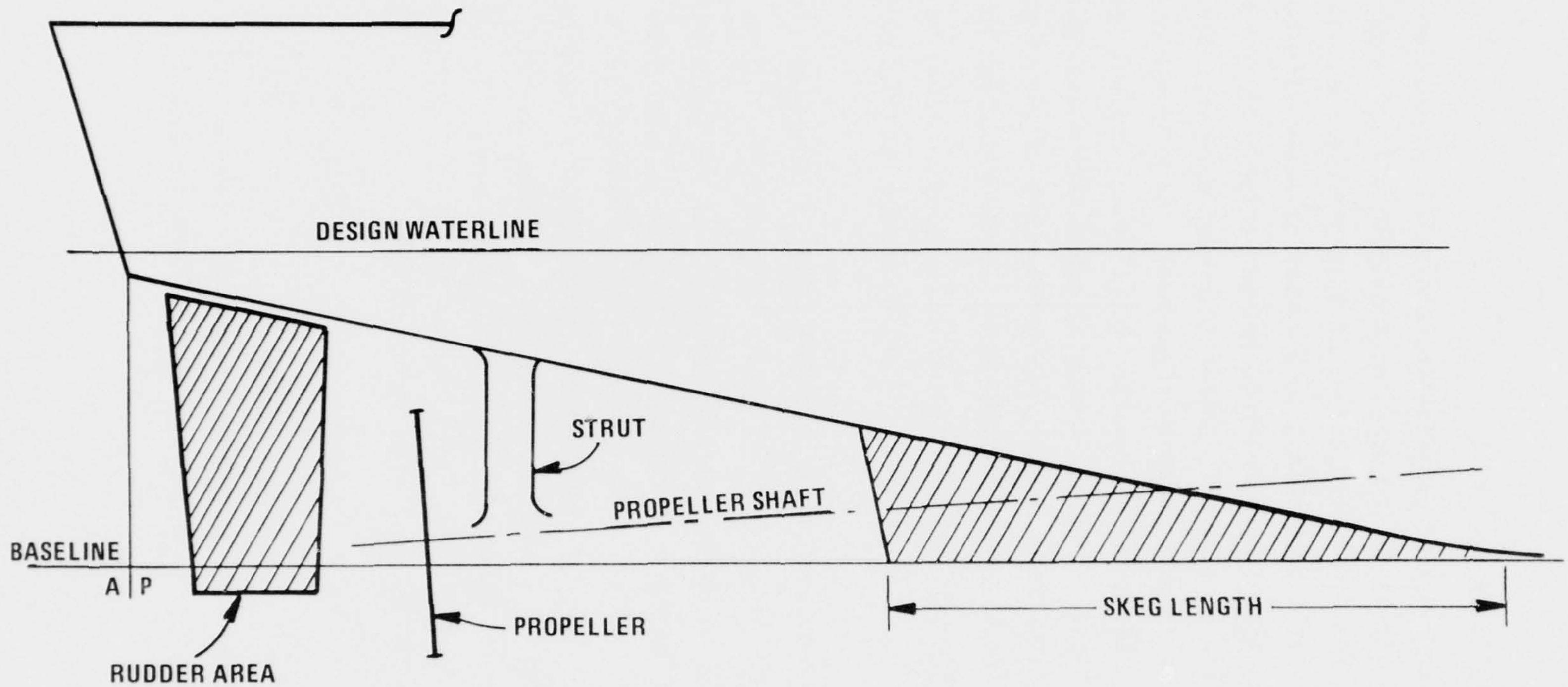


Figure 4. Rudder/Skeg Sizing

2. Model Test Scale Effects

Due to the lack of satisfactory analytical evaluation techniques, model testing is relied upon in developing a final appendage configuration and for final predictions of ship performance. Stability and control model tests are subject to a number of scale effects. Studies have shown that the correlation between actual ship performance and model prediction is generally better than should be expected from the number of scale effects present. Apparently, some of the principal scale effects tend to cancel one another for typical ship hull and appendage configurations. There is a lack of quantitative information on the individual effects on predicted ship performance of the several principal scale effects. Such information is needed in order to:

a. Permit valid corrections to be applied to model tests results, especially those for unusual hull/appendage configurations, and

b. increase confidence in predictions of ship stability and control characteristics based on model test results.

As an indication of the cumulative effects of the scale effects mentioned above, Figure 5 is presented. This figure, taken from Reference 4, compares tactical diameters predicted from model tests with full scale values. The data represents a large number of ship designs tested at the Admiralty Experiment Works (AEW) in Great Britain. It appears that the model tests, on the average, over-predict tactical diameter by about 10 per cent. This conclusion is, of course, dependent upon the particular model test techniques employed.

Seakeeping

General Approach

In the Conceptual Design Phase, seakeeping characteristics are generally assessed but not in detail. Usually some measure of overall

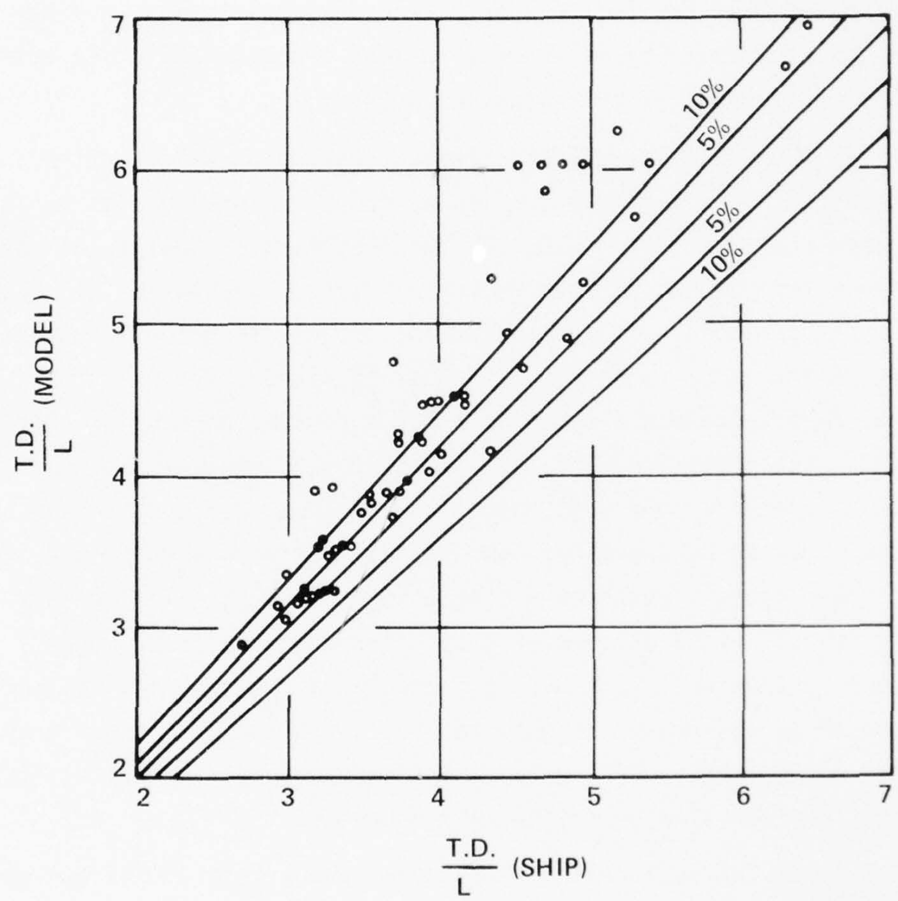


Figure 5. Ship/Model Tactical Diameter Comparison

seakeeping performance pertinent to the particular design in question is developed and then each alternative design study evaluated in terms of the measure. In this manner, relative seakeeping performance is addressed in the overall assessments of ship performance made in developing the Navy Requirements for the new ship's design. Failure to do this in the past has led to some bitter disappointments when new ships have gone to sea. Seakeeping evaluations are generally made using highly simplified methods at this stage of design.

In the Preliminary Design Phase, seakeeping considerations play an important role in the selection of the principal hull dimensions and form characteristics. Due to the present availability of computer programs for evaluating ship motions in irregular seas at all headings, it is now possible, within limitations, (much of what follows discusses these limitations) to develop a number of alternative hull forms and assess their motion characteristics as a routine part of the design optimization process. Certain effects of ship motions, such as deck wetness, slamming, and resistance in head seas, can also be evaluated. On the other hand, there are some seakeeping factors which are important in certain designs which cannot be assessed analytically at this time, e.g., the effect of abovewater knuckles on deck wetness. Once a Preliminary Design hull form has been developed, studies are routinely conducted to determine the need for roll stabilization other than the normal bilge keels. If a need is evidenced, then studies to select system type and size (capacity) are conducted.

In the Contract Design Phase, seakeeping model tests are generally conducted. These tests typically have four goals:

1. To confirm previous analytical performance predictions.

2. To assess those critical aspects of performance which available analytical techniques cannot.

3. To provide data required by certain subsystem designers, e.g., sponson slamming loads for the structural designers.

4. To optimize specific hull form features.

The final hull form is established and the roll stabilization system design is completed. It should be noted that the first three of the four test goals listed above pertain to the stabilization system design and performance as well as to the ship hull design and performance. Final predictions of seakeeping performance are made at the end of the Contract Design Phase. These predictions are based largely on model test results.

Specific Problems

1. Deck Wetness and Slamming

These two principal motion effects will be discussed together rather than separately. At present, extensions of computer programs for predicting motions in head seas are used to assess these two phenomena (the frequency and severity of weather deck wetness and keel slamming) for the ship's hull prior to model testing. More refined analytical prediction techniques are required. For evaluation freeboard adequacy and deck wetness characteristics, the effects of ship roll, the ship bow wave in calm water, wind speed and direction, and the abovewater hull shape must be considered. With regard to slamming, present techniques must be extended to address what has been called "flare slamming": slamming of the ship hull sides. For both keel and flare slamming, prediction methods must account for ship roll which is ignored at present. Although it is recognized that the phenomena being discussed here are extremely complex, improved analytical evaluation methods must be developed before rational studies of alternative hull forms from the standpoints of slamming and deck wetness can be conducted. Model testing to evaluate hull form alternatives from these

standpoints is out of the question in most cases due to time and cost constraints. When such improved analytical evaluation techniques have been developed, comprehensive studies of the effects of alternative abovewater and below water section shapes on deck wetness characteristics and slamming frequency and severity (local and hull girder loads) will become feasible.

Another important aspect of wetness and slamming concerns abovewater hull appendages such as gun sponsons and deck edge aircraft elevators. Figure 6 shows a typical sponson. When a design incorporating such appendages is being developed, tentative decisions regarding minimum acceptable height above water (freeboard) must be made at an early date. Later, decisions regarding sponson shape must be made. For scantlings to be developed, predicted wave impact forces are required. Although model testing can be used to assess such factors, in most cases model tests are not practical due to time and cost constraints. Frequently, feasibility studies are requested which must be completed in a matter of days and decisions regarding minimum acceptable freeboards must be made quickly. Improved theoretical or empirical methods are urgently needed. At present, naval architects are attempting to deal with such issues using existing computer programs for predicting ship motions in head seas. This is clearly unsatisfactory. Such programs ignore ship roll, do not adequately account for the complex interactions between the ship's hull, its calm water wave train, and the encountered ocean waves, and do not permit sponson impact loads to be assessed. One only needs to stand for a few minutes on the lowered forward-most aircraft elevator on an aircraft carrier steaming at sea to gain an appreciation of the complexity of these interactions!

2. Roll Motion Predictions

Of the six motion components, roll is probably the single most important from the standpoint of its effects on ship performance

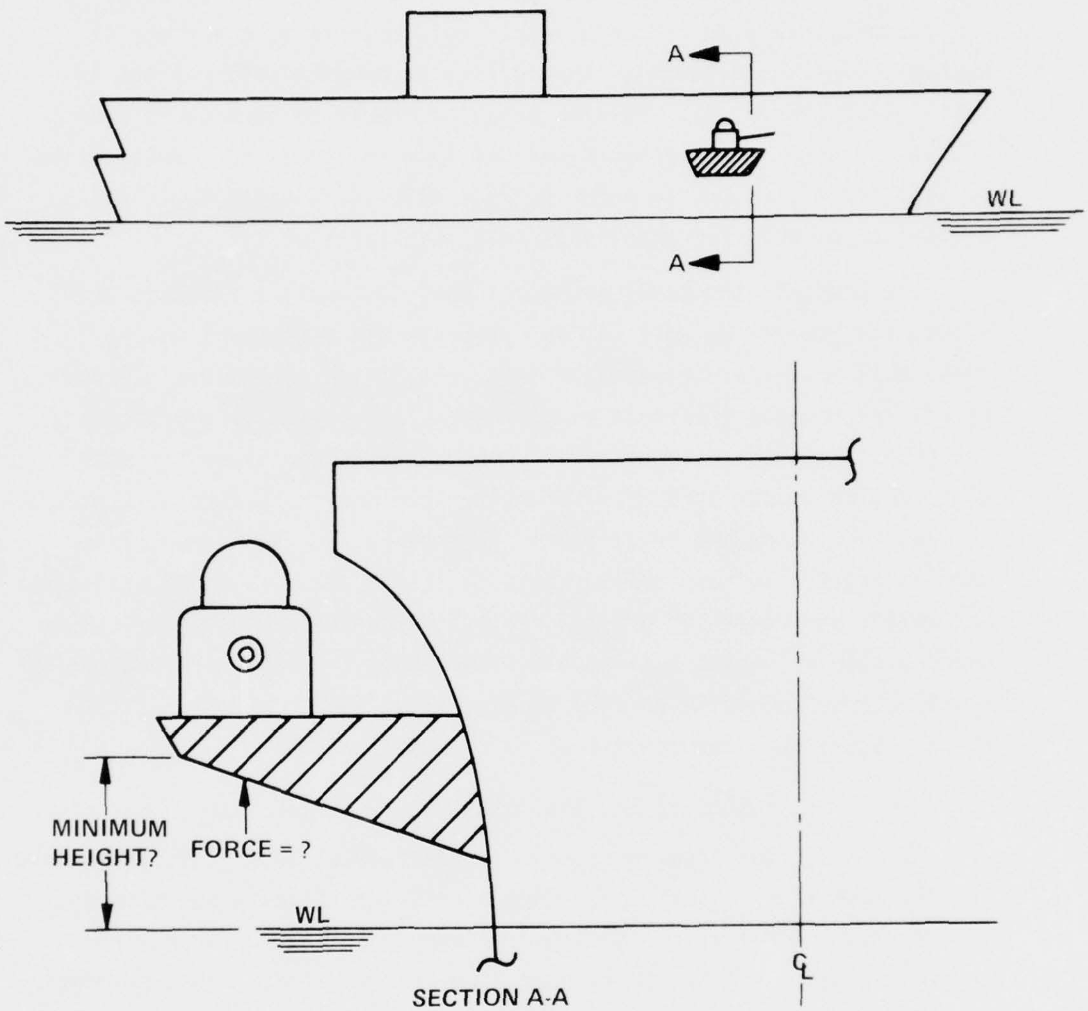


Figure 6. Abovewater Hull Appendage Sketch

specifically, crew, weapons and sensor performance. Fortunately, it is relatively easy to reduce a ship's roll motions by a variety of methods. For these reasons, the ability to predict roll motions is particularly important. Rolling behavior should be considered when a new ship design's hull proportions and form are selected. Roll motion predictions are needed in order to rationally size bilge keels and to determine the need for additional roll stabilization.

At present, the naval architect does not possess adequate techniques for predicting roll motion. Easy-to-use methods of making reasonably accurate estimates of ship roll motion are needed. Actually, several techniques are required of increasing complexity and accuracy. The crudest method would be used in the early design stages to make gross motion comparisons of alternative ship design studies with widely varying hull sizes and proportions. A more refined method would be used in the Preliminary Design Phase to assess the effects of hull form and weight and center of gravity variations on rolling behavior. Once the Preliminary Design hull form was selected, the most accurate prediction technique would be used to size bilge keels for the hull and to assess the need for additional roll stabilization.

3. Roll Stabilization System Evaluation

A current problem relates to the evaluation of the performance of roll stabilization systems. Once a roll stabilization system has been designed for a ship, final performance predictions for the ship with and without the system in operation are required. These predictions are particularly important for stabilization system performance since the cost and uncertainties associated with full scale performance demonstration trials in this case make such trials unsuitable for routine confirmations. In other words, the final performance predictions in this case are not likely to be supplemented by full scale trial data. There are problems associated with all of the available techniques for

making such performance evaluations: analytical, experimental, and combinations of the two. Some major problems result from the need to predict performance in quartering seas when, typically, wave encounter frequencies approach zero. Other problems result from the nonlinearities of the phenomena involved, the scale effects associated with model stabilizer fin and bilge keel lift forces, and the strong coupling between ship roll and rudder angle in quartering seas. Improved methods of evaluating the performance of stabilization systems, after their design is complete, are required. Such methods should permit the realistic evaluation of ship motions with and without stabilization for a wide range of ship speeds and headings in short crested irregular seas as well as in long crested swells.

4. Powering in Waves

The ability to predict a ship's powering characteristics in irregular seas is required. This is especially true for large ships whose speed in rough water is more frequently limited by installed shaft horsepower than by ship motion effects. This is another instance where model testing, although possible, is out of the question for comprehensive evaluations due to time and cost considerations. Research to date has resulted in practical methods of predicting added resistance in irregular head seas. The major remaining obstacle is the prediction of propulsive coefficient effects. Additional research is required in order to develop reliable methods of predicting the effects of rough water operation on the several components of the propulsive coefficient: wake fraction, thrust deduction factor, and relative rotative efficiency.

Propeller Induced Vibrations

A rotating ship's propeller transmits vibratory forces to the ship's hull and propulsion machinery through the propeller shaft and the water surrounding the propeller. The forces transmitted excite vibrations, both of the hull girder acting as a beam and of local structural

panels and machinery components. Ship vibration is becoming a matter of increasing concern as propulsion horsepowers rise. The naval architect is in great need of additional guidance to permit the design, with confidence, of ships with acceptable vibration levels. Two problems, closely related, will be mentioned.

In the Preliminary Design Phase, when the preliminary appendage configuration is being developed, guidance regarding the minimum acceptable clearances between the propeller and adjacent hull and appendage surfaces is needed. These include the propeller tip-hull clearance, the propeller-rudder clearance, the propeller-strut clearance, and the propeller-skeg clearance (see Figure 4). At the present time, only the crudest "rules of thumb" based upon past experience are available. These rules do not reflect variables whose influences are known to be primary. Recent research has resulted in valuable pieces of information for specific ship designs but no general guidelines have been developed.

Once the ship design has progressed to the point where the primary hull structure has been tentatively defined along with other features such as the preliminary appendage configuration, the hull form, the primary ship arrangements, etc., a vibration study can be initiated. This generally occurs at the beginning of the Contract Design Phase. At the present time, the weak link in such studies is the estimate of the vibratory forces transmitted to the ship's hull and appendages (strut and rudder, for example). Present techniques for estimating such forces correlate poorly with model measurements. Furthermore, this problem can be greatly complicated when propeller cavitation occurs as is frequently the case at the higher ship speeds. Improved, comprehensive, and relatively easy-to-use methods of predicting propeller generated vibratory forces are required. The methods should permit the individual component forces to be estimated (rudder, strut, hull, etc.) as well as the net force resulting from the combined forces. Propeller

cavitation effects must be accounted for. Inputs to the evaluation would include hull, propeller, and appendage geometries and the estimated ship wake field "seen" by the propeller.

Other

Mentioned briefly below are two miscellaneous phenomena for which present prediction methods are inadequate.

1. Trim and Sinkage (in calm, deep water)

Prediction methods are required to assist in seakeeping evaluations (deck wetness and slamming) prior to conducting model tests. Calm water trim and sinkage at ship speeds greater than zero must be taken into account in deck wetness and slamming studies due to the influence of these two factors on freeboard and draft forward.

2. Stack Gas Flow

It is important to be able to predict stack gas plume trajectories and temperatures for a wide range of ship operating conditions (relative wind speed and heading, propulsion plant power level, etc.). Stack gases affect aircraft operations, adjacent weapons, sensors and other equipments, performance of crew when on deck, and the maintenance of topside surfaces.

Semi-empirical formulae have been developed for making such predictions. The formulae are quite satisfactory for "free stream" situations where the effects of hull, superstructures, masts, etc. can be ignored. Figure 7 compares a predicted plume trajectory with full scale trial data. The equation noted in the figure and used to make the prediction was derived from Reference 5.

At present, model tests must be relied upon to assess plume trajectories in situations where hull, superstructure and similar effects cannot be ignored. Typically, such tests are slow and expensive. The test results are difficult to interpret (the primary means is by visual

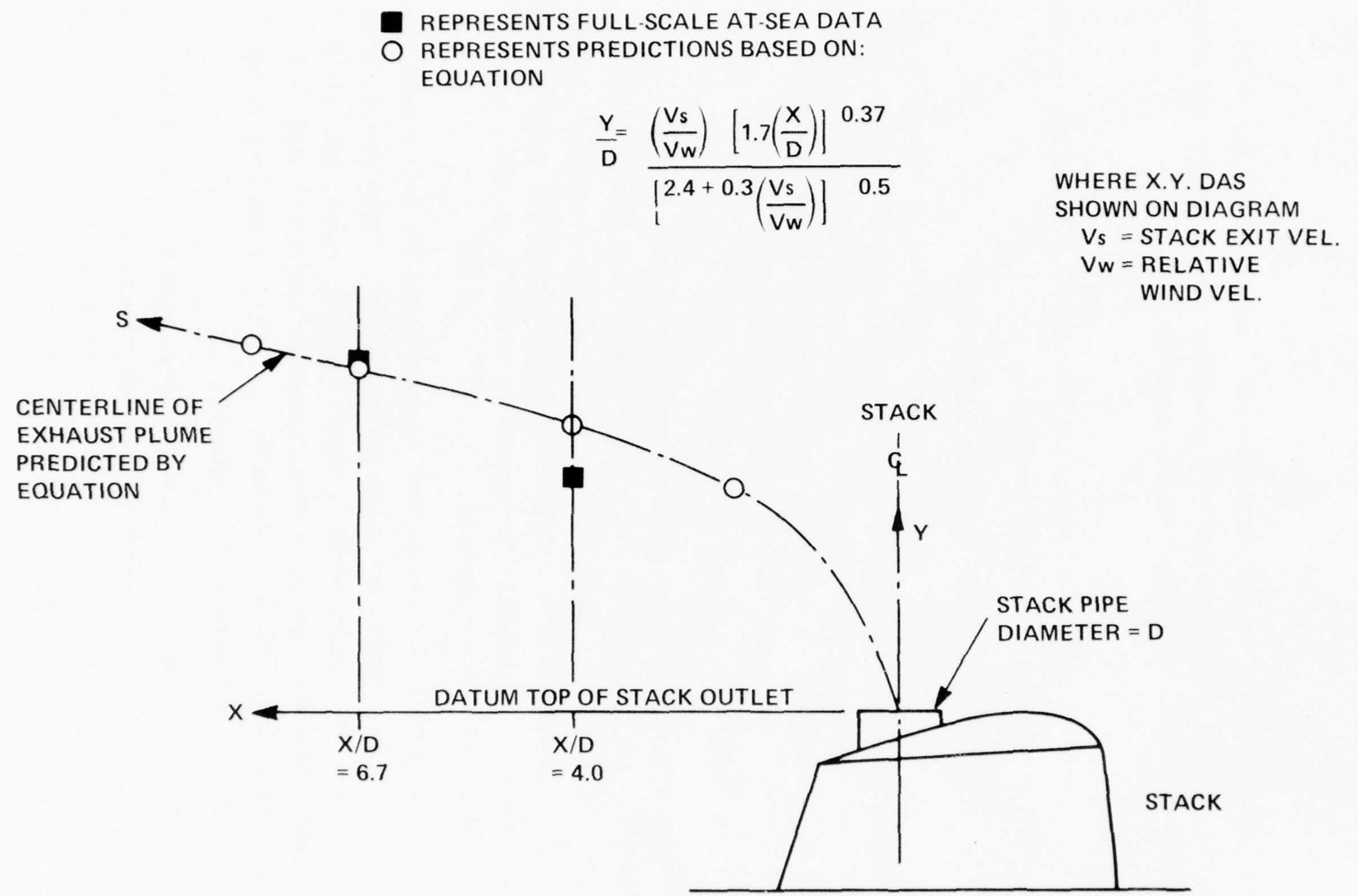


Figure 7. Plume Trajectory Comparison

observation of the tests) and the effects of stack gas buoyancy are generally not accounted for. A valid analytical prediction technique for such cases would be of great value in evaluating the relative merits of alternative stack, superstructure, and mast configurations.

Summary

Improved prediction methods are required in the problem areas discussed earlier. In some cases these methods are needed in order to ensure satisfactory ship performance such as predicting propeller induced hull pressure forces and roll stabilization system performance in quartering seas. In other cases, the improved methods are needed in order to efficiently assess the performance of a large number of alternative configurations and select the "optimum" one rationally. For example, predicting bare hull resistance falls into this category along with predicting the components of the propulsive coefficient other than the open water propeller efficiency.

HIGH PERFORMANCE SHIPS

Introduction

Within the class, High Performance Ships, have been included:

- SWATH - Small Waterplane Area Twin Hull, Figure 8.
- SES - Surface Effect Ship (hard sidewalls), Figure 9.
- ACV - Air Cushion Vehicle (full peripheral jet, fully skirted), Figure 10.
- Hydrofoil - Figure 11.

They are called high performance because they all either have or are expected to have dramatically better performance than conventional monohulls in some particular characteristics. In the case of SWATH (variously called S^3, LWP, MODCAT, and others), the seakeeping and speed are not expected to be significantly degraded until very severe sea states are reached. However, smooth water performance will probably be a bit poorer than a conventional monohull's.

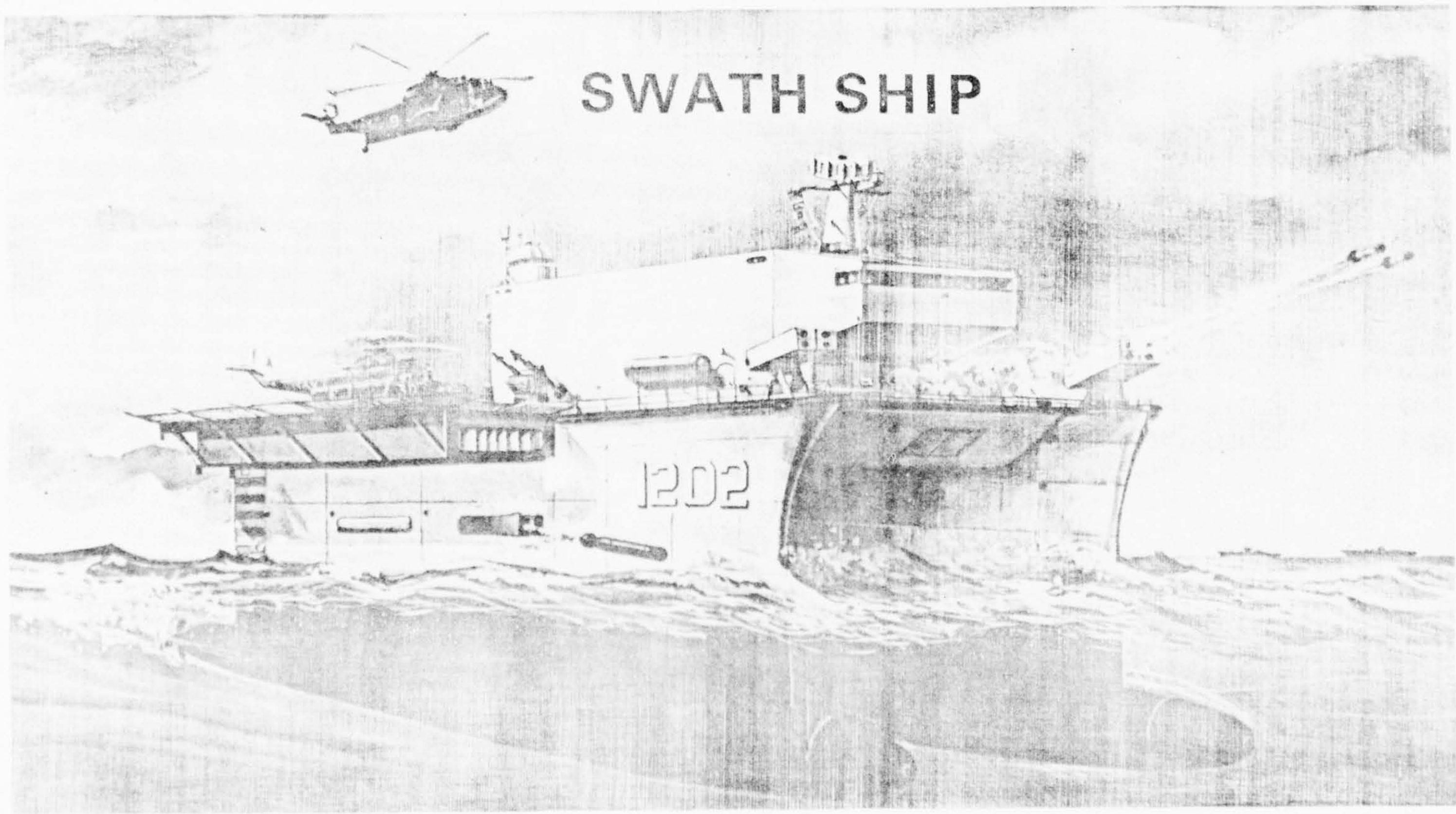


Figure 8. Artist's Concept of a Possible Swath (Small Waterplane Area Twin Hull) Ship

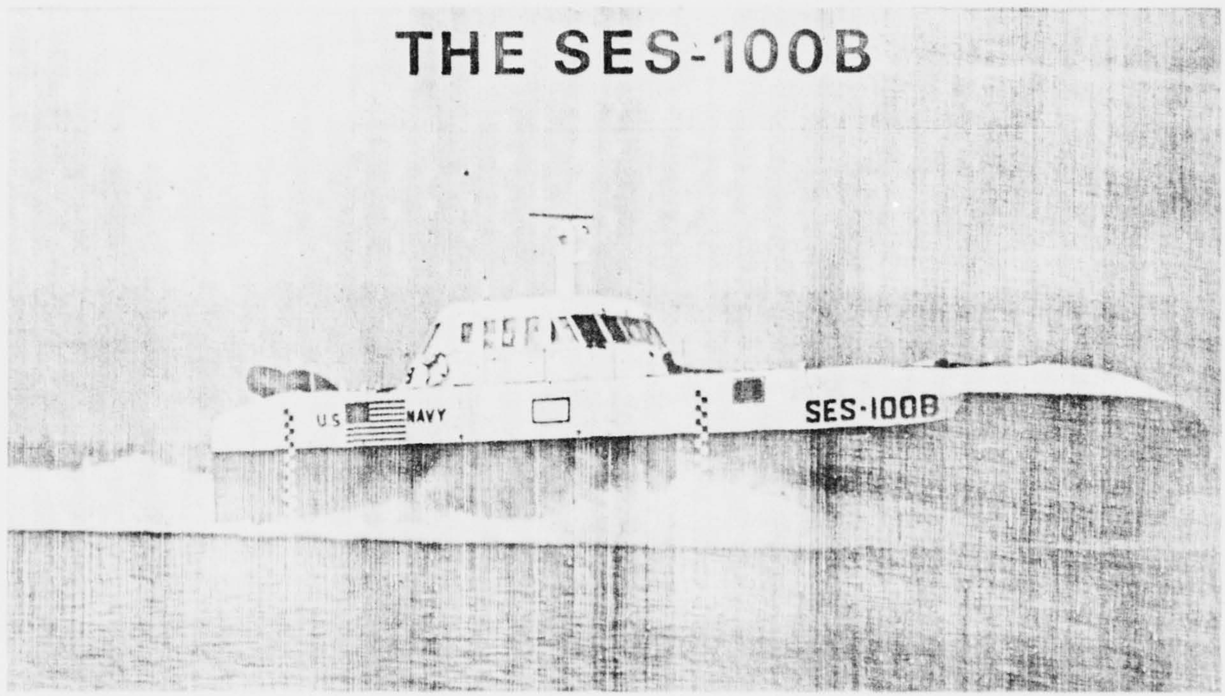


Figure 9. The SES-100B

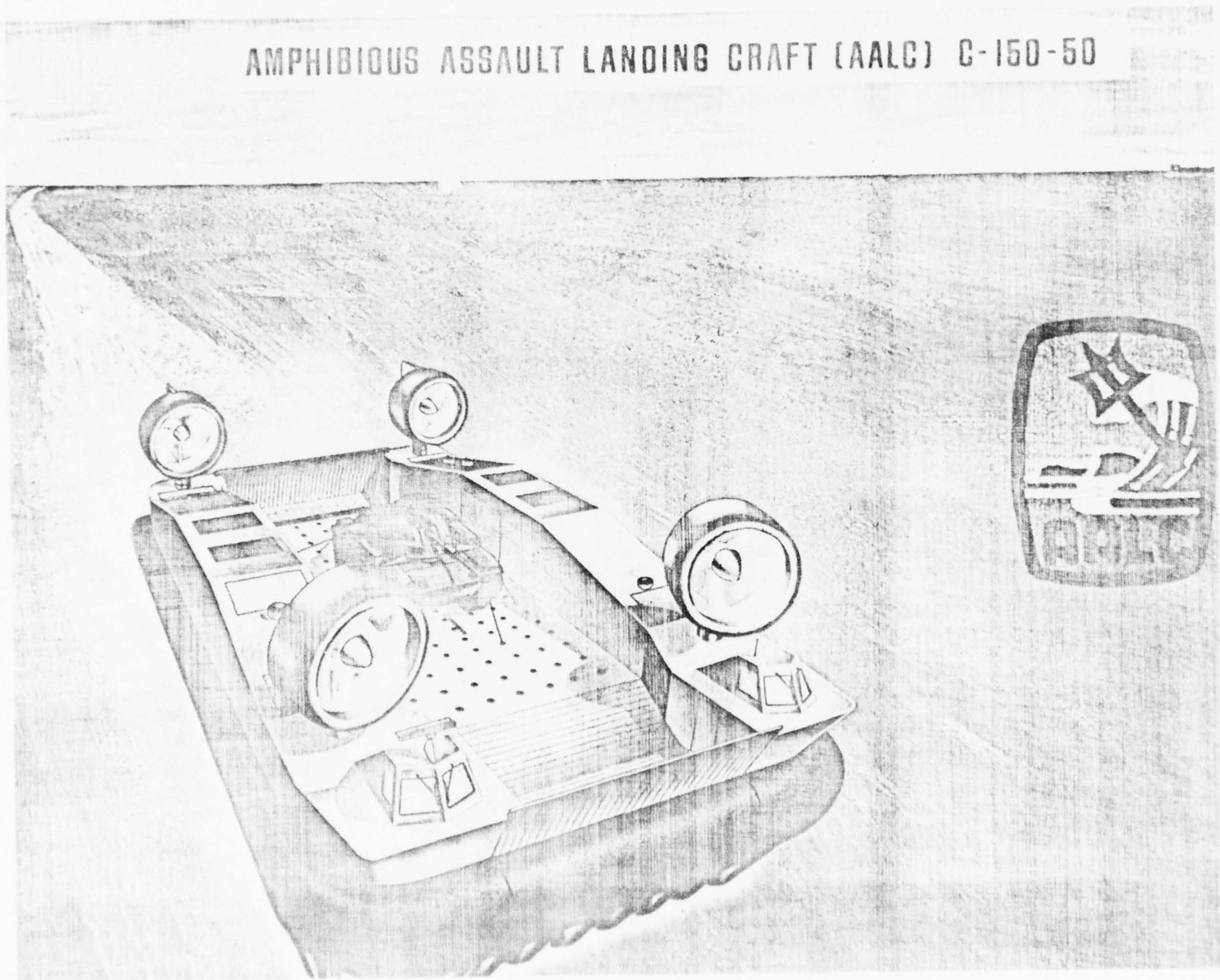


Figure 10. Artist's Concept of the AALC (Amphibious Assault Landing Craft) Program's C-150-50

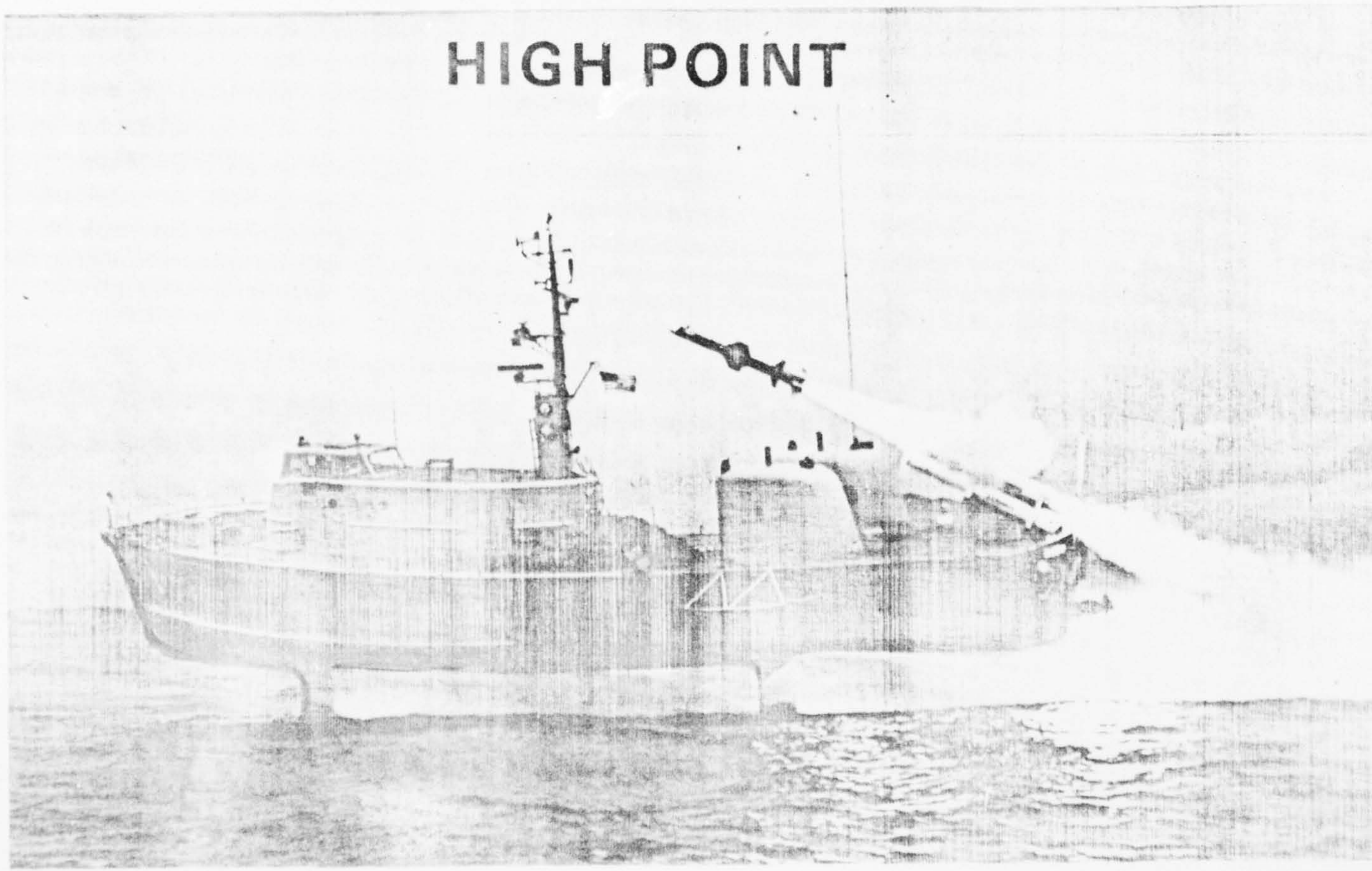


Figure 11. A Harpoon Missile Being Launched from High Point (PCH1)

The subcavitating hydrofoil will probably not find applications for large sizes or very high speeds. Because the open ocean seakeeping performance of the fully submerged hydrofoil is much better than the surface piercing hydrofoil, the succeeding discussions will be limited to this type of foil. The supercavitating (or ventilated) hydrofoil is not of current interest for lift systems. However, research in this area is relevant to two other problems: control surfaces for other high performance ships, in particular the SES, and as fundamental work supporting supercavitating propeller designs. The SES is of interest because of its potential for higher speed and larger sizes than any of the other high performance ships. The amphibious capability of the ACV puts it in a different category from the others in that it can compete where the mission requires very rapid transit over the beach or highly heterogeneous terrain, e.g., tundra, swamp, or any limited navigational draft.

While these different platform concepts appear to have different regions of applicability in terms of size and speed, they have two things in common with respect to design tools development:

1. There is little or no design data and full scale experience available to the U.S. Navy in the area of hydrodynamic performance with the possible exception of the hydrofoil, and

2. their higher performance merits more sophisticated design in order to get the most for the investment, i.e., more accurate prediction techniques are needed, and should be applied earlier in the design process.

Propulsion

In this section and in the following two sections the different platform types will be handled separately.

Drag

1. SWATH

Prediction of resistance for SWATH forms generally follows conventional monohull practice. Frictional and residuary resistance are separated and scaled using Reynolds and Froude Numbers, respectively. Three methods for predicting the wave resistance part of the residuary resistance using classical thin ship theory have been developed by Pien,[6] Chapman,[7] and Lin.[8] All three methods utilize a sheet of sources to represent the strut and a line source distribution to represent the hull. None of them allow a variation of source strength as a function of depth for the struts. Pien makes provision for using both a vertical line source and/or doublet to represent a bulb in the strut.

Pien starts with singularity distributions and finds the hull which has good wave resistance properties. Chapman needs a section area curve of the hull and an analytical representation of the strut to start, and Lin's method can accept offsets. Chapman's method has been found very useful in early stage design. Pien's method is more cumbersome to use, but has been used by a NAVSEC naval architect, and is most useful for research and optimization studies. All three methods can account for the curved flow field induced by each hull/strut(s) combination on the other. However, the resulting cambering is very small and has, therefore, not been used for construction reasons.

All three methods compare favorably with experimental data, which is not surprising in light of the slenderness of the SWATH hull/strut(s). Figure 12 [9] shows a comparison of theory and experiment for Lin's and Chapman's theories. Figure 13 [10] is a comparison of Model 5301 (SWATH V) with Lin's theory. Figure 14 [6] is a comparison of Pien's wavemaking resistance prediction with measured residuary resistance.

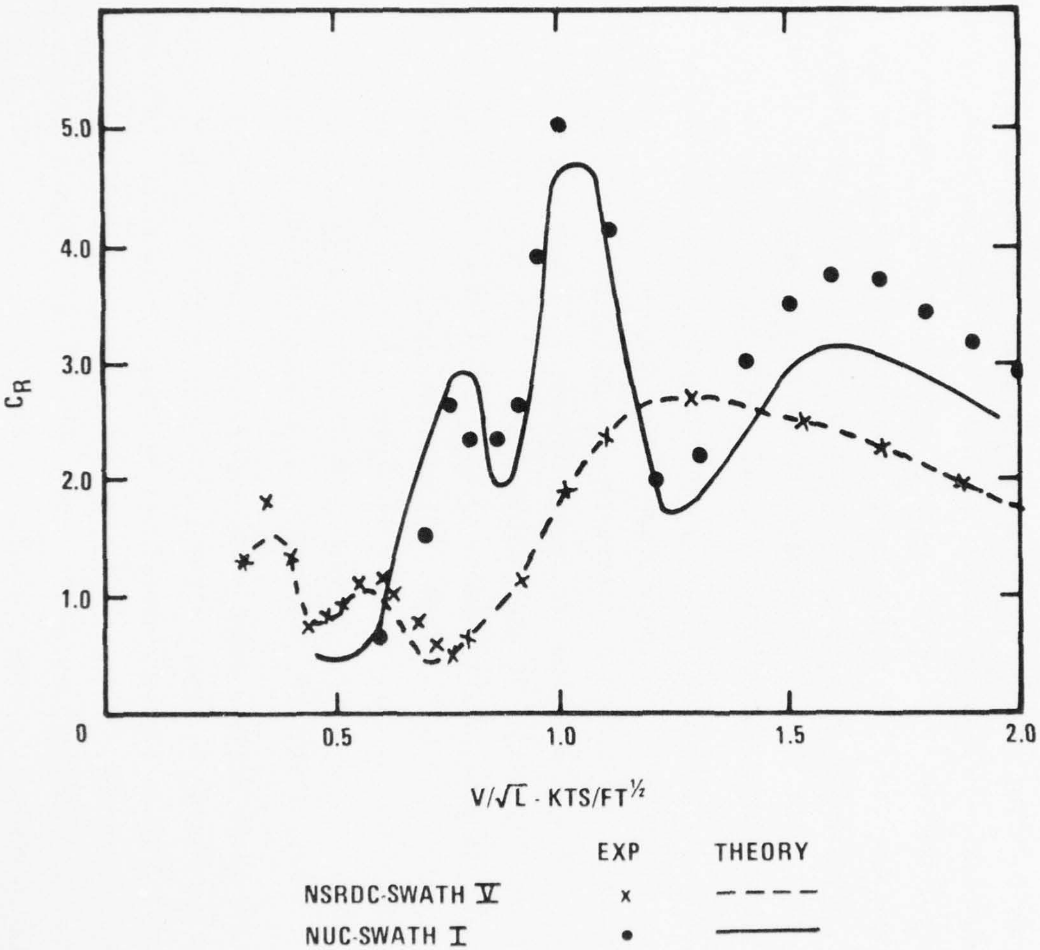


Figure 12. Comparison of Lin's Theory (Dashed) and Chapman's Theory (Solid) With Experimental Data

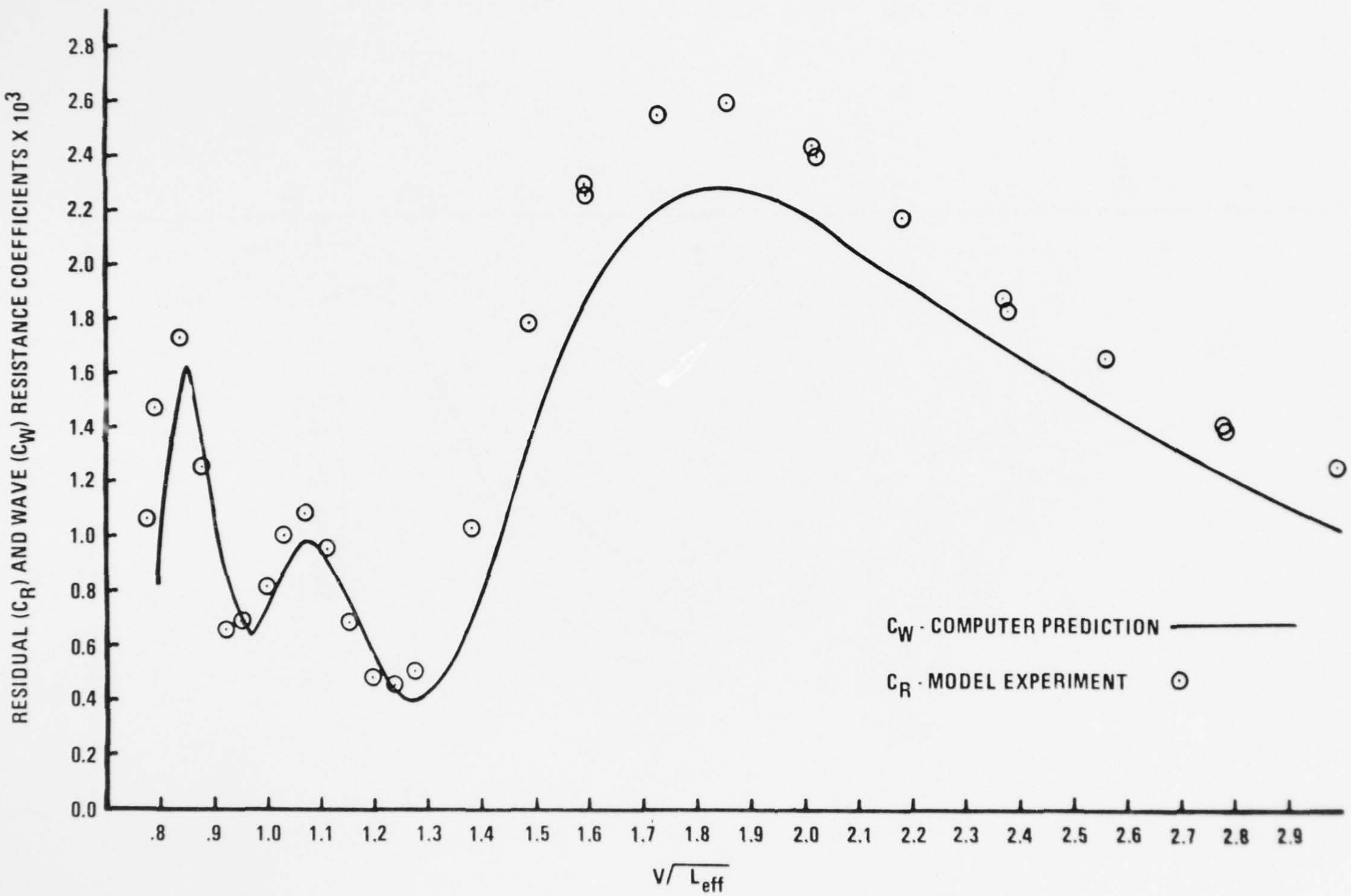


Figure 13. Comparison of Lin's Theory With Experiment

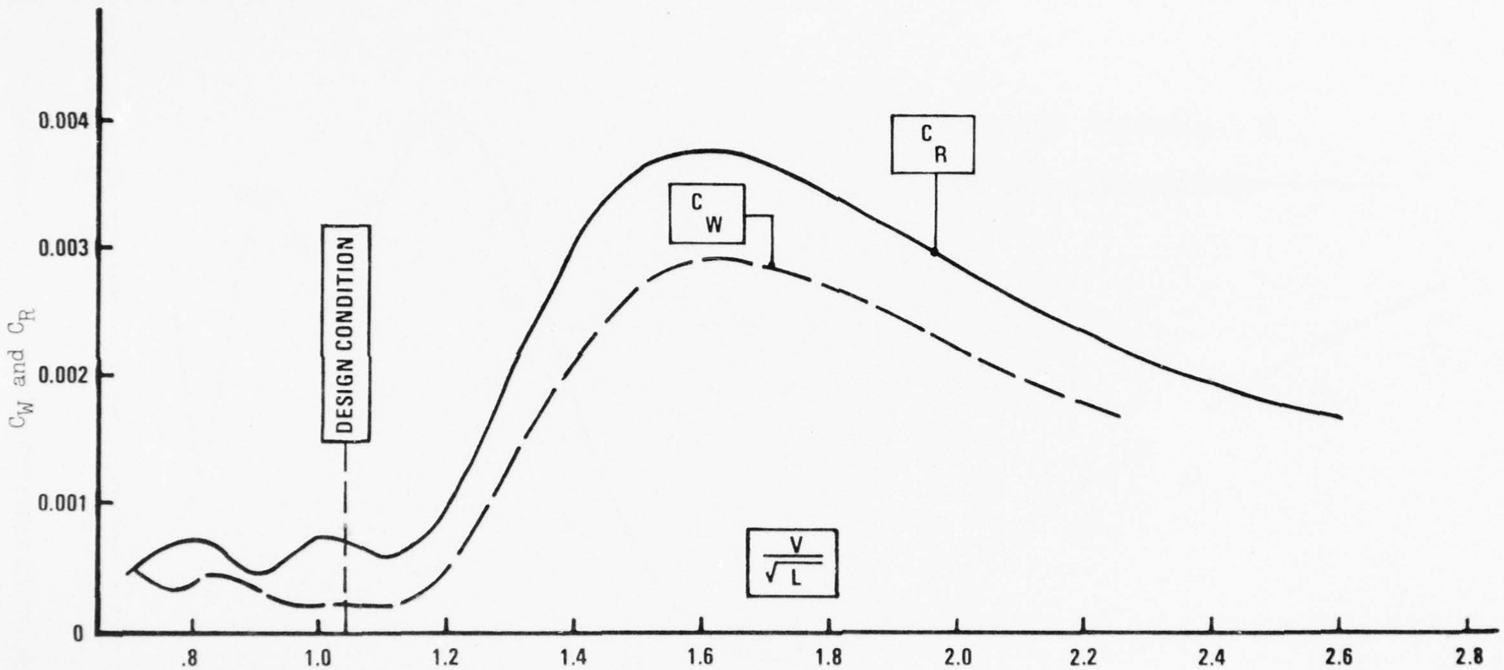


Figure 14. Comparison of Pien's Theory, (Cw) With Residuary Resistance, (Cr)

Other drag components considered are:

· Frictional, based on wetted surface and Reynold's number based upon the length of the hull component.

· Form drag, which has been taken as zero in current studies because all submerged components can be nicely faired.

· Spray drag which is discussed below.

· Interference drag between fore and aft struts which is discussed below.

· Aerodynamic drag of the abovewater geometry which is not large and so has been computed using a drag coefficient of 1.0 based on frontal area.

· *Spray drag has been* investigated experimentally by Chapman.[11] However, he leaves some unanswered questions; one is with respect to how the spray drag varies with Froude Number, and indeed if this is how it should be scaled. A second is with respect to whether his data is applicable at the very high speeds associated with hydrofoils and SES's. Figure 15 shows his results for a strut of thickness/chord ratio (t/c) of 0.15, a chord of 18 inches, and at three submergences, d.

Equation 7 referred to in the figure is

$$\frac{D}{q} = .011\ ct + 0.08t^2$$

where q is the dynamic pressure. The whole subject of strut drag, ventilation inception, and cavitation inception is one for which the ship designer needs considerable help for all high performance ship types.

Interference drag between fore and aft struts is currently an anomaly. It is also moot with respect to proposed designs since only one strut per side is being recommended. Using Chapman's method,[7] the

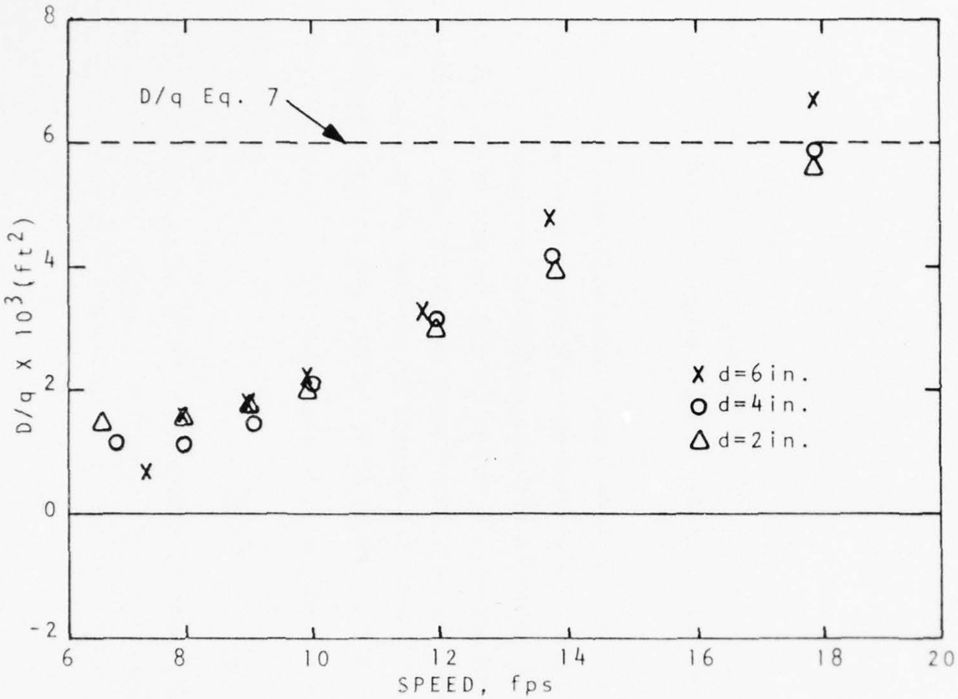


Figure 15. Theoretical and Experimental Wave-Making Drag Coefficient For Surface Piercing Struts by Chapman

effect of gap-chord ratio has been studied analytically and reported in Reference 12. Figure 16 [12] shows that reducing the gap is beneficial. But Figure 17, which is unpublished experimental data due to G. Dobay of the Naval Ship Research and Development Center, shows that reducing the gap is deleterious. Some even more recent experimental work implies that reducing the gap may or may not be deleterious depending on other parameters, e.g., Froude Number. Currently, SWATH design studies involving multistruts use zero as the estimate of this drag in light of the above situation.

2. ACV/SES

The drag is separated into:

- Wave resistance of the cushion
- Aerodynamic drag of the abovewater body
- Seal drag
- Sidewall drag (if present)

In addition, venting of the cushion in a seaway and added resistance in waves must be allowed for. Momentum drag is included in the propulsive coefficient for accounting purposes. Both the seal and sidewall drags must be further separated into friction, spray, form, and even wavemaking components.

The wave resistance of the cushion is computed using Newman and Poole's[13] formulation and this appears to be in satisfactory agreement with experimental data, see Figure 18 from Reference 14. An approximation to it has been developed by Chaplin and Ford[15] and the exact theory has been reduced to a standard computational procedure by several users, e.g., R.F. Bjorkland of NAVSEC has written an undocumented FORTRAN program to execute it which includes the low Froude Number region and thereby accounts for take-off drag. The aerodynamic drag has been treated only empirically for these craft. The most sophisticated approach has been to build up an overall drag by separately considering

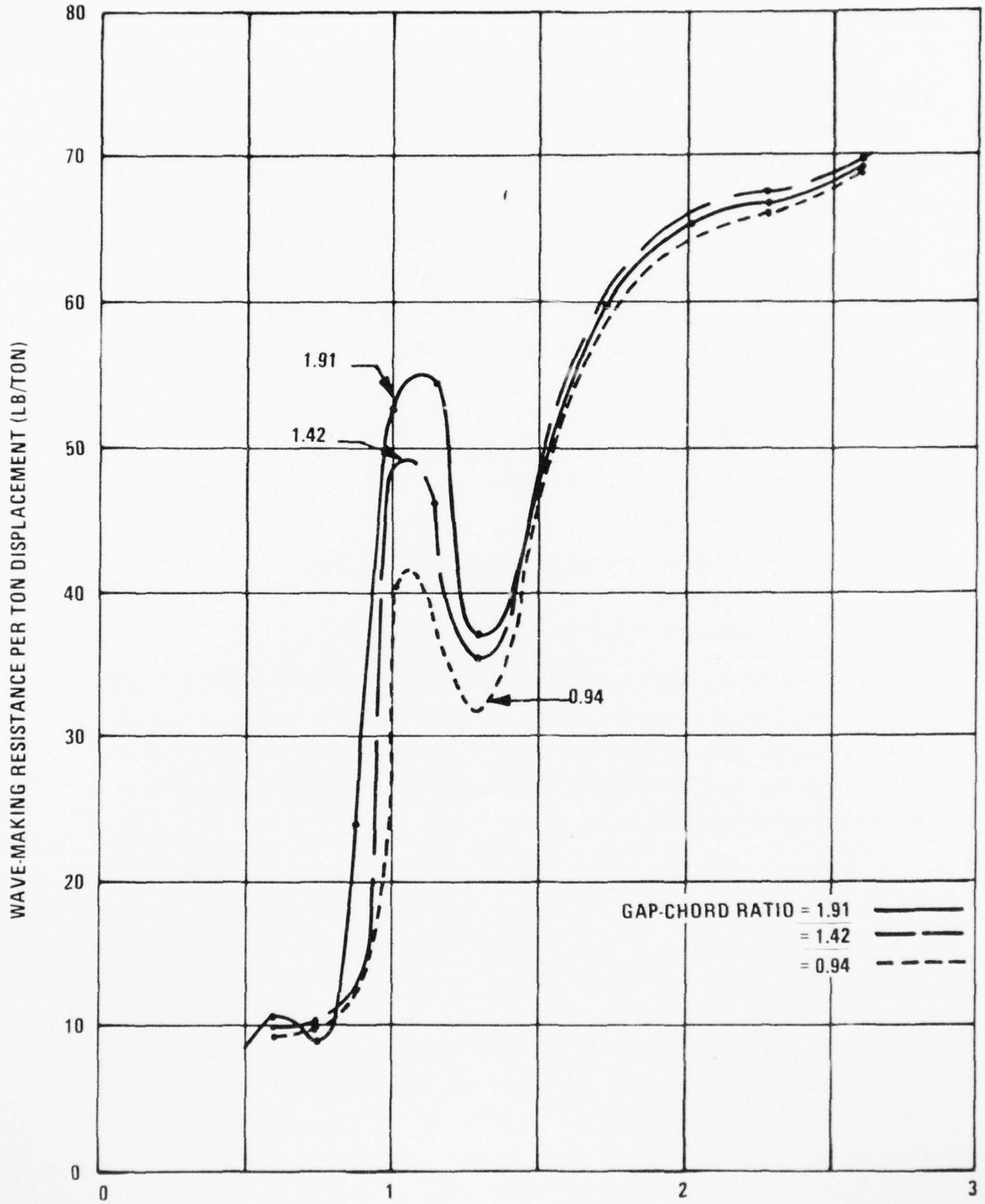


Figure 16. The Effect of Strut Longitudinal Spacing on Theoretical Wave-Making Resistance as a Function of Speed-Length Ratio (Froude Number)

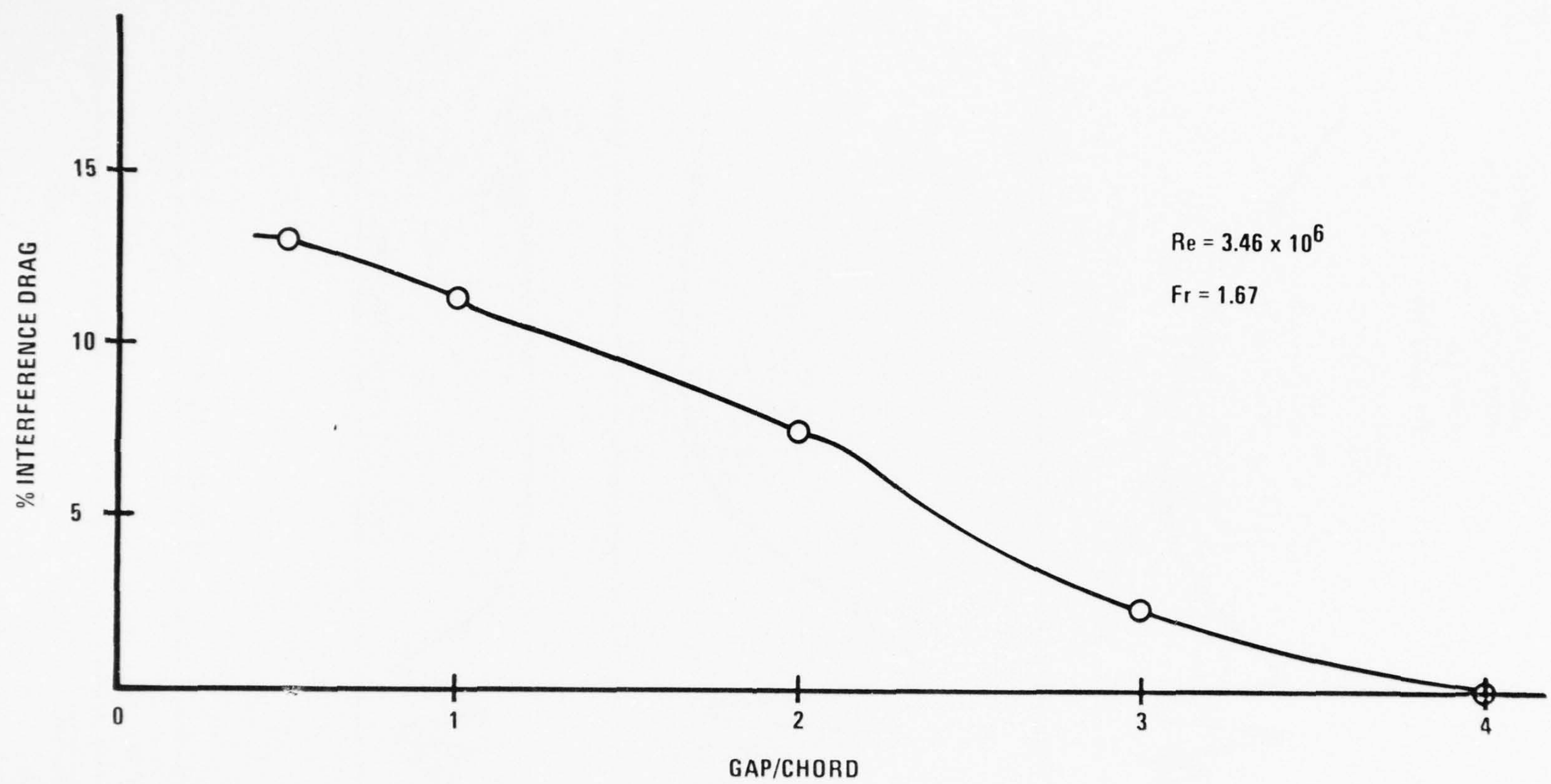


Figure 17. An Experimental Result Showing the Effect of Strut Longitudinal Spacing at One Froude Number

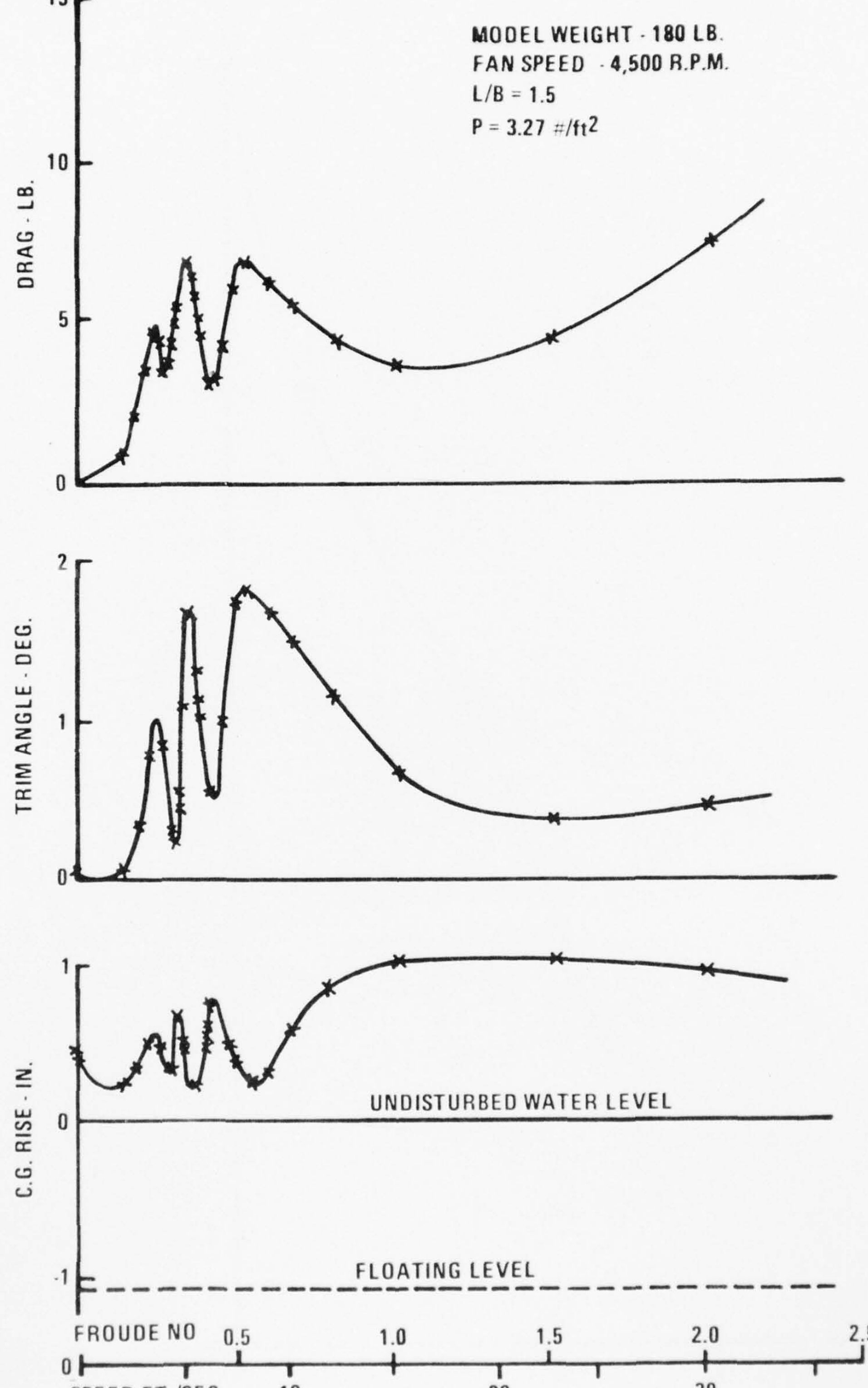


Figure 18. A Comparison of Newman and Poole's Theory for Wave-Making Resistance of ACV's With Experiments by Everest and Hogben

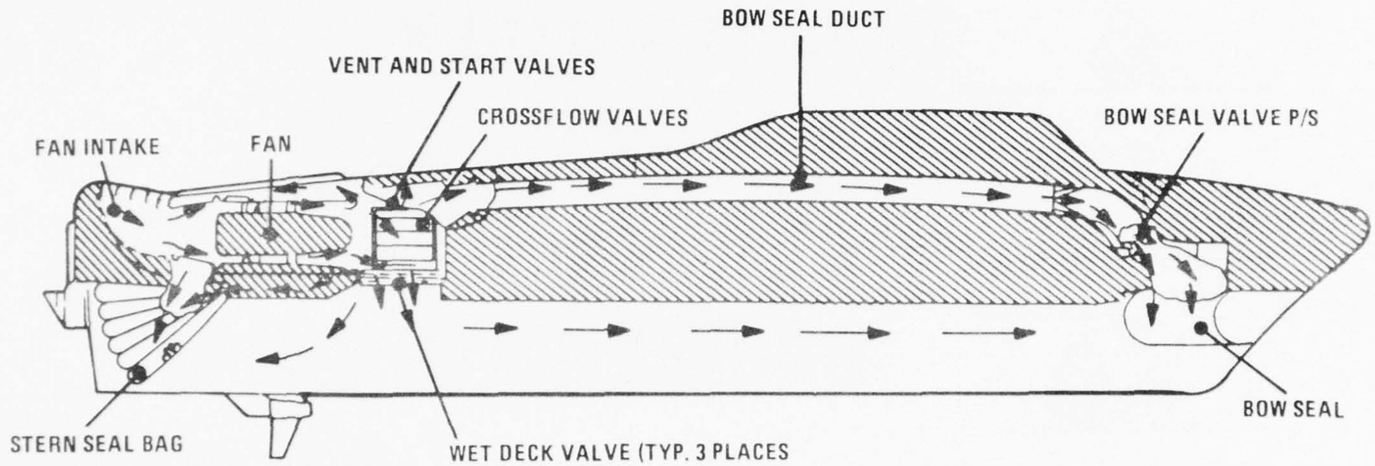


Figure 19. A Section Through A Typical SES Illustrating the Airflow System

Figure 20. Possible Seal Types

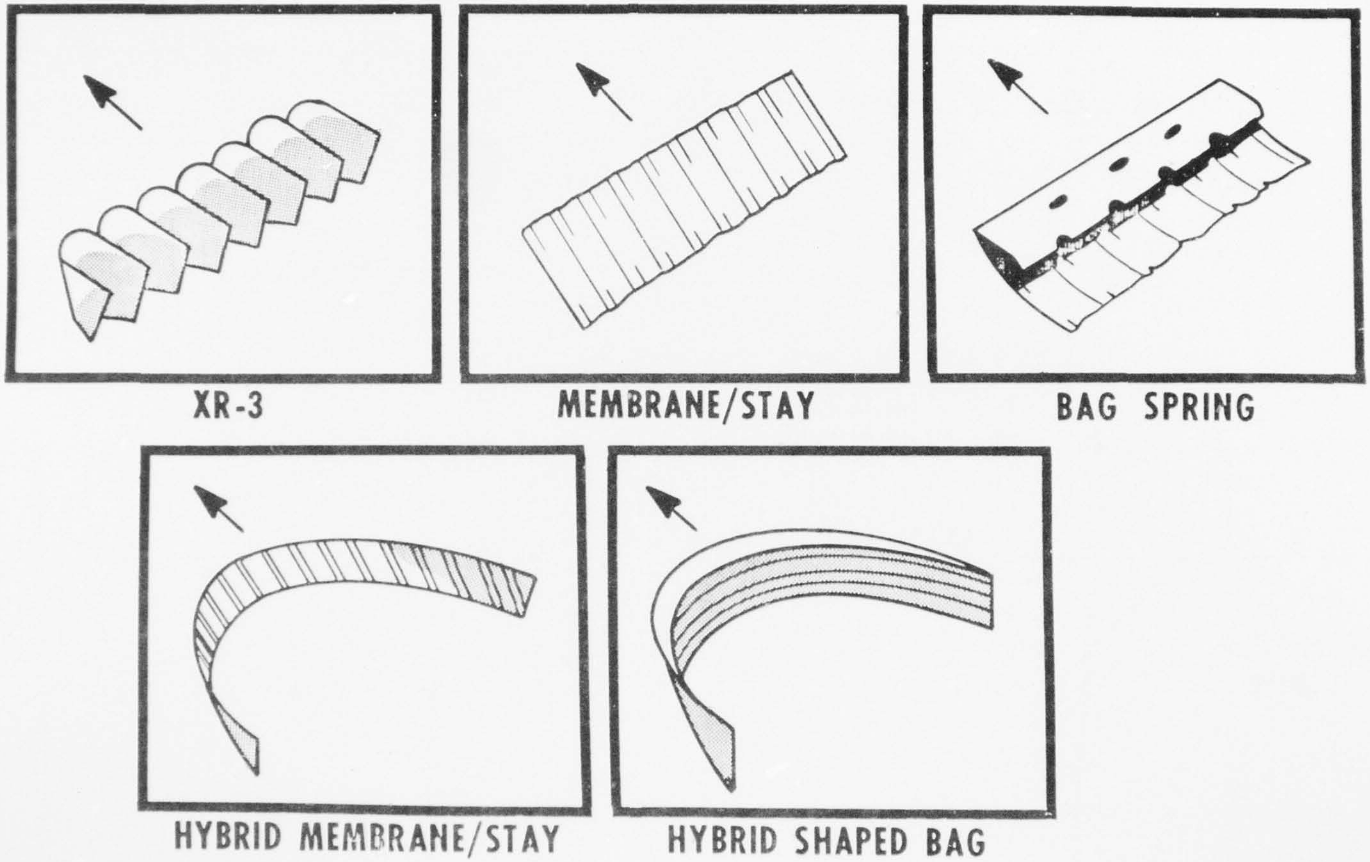


Figure 21. Possible Types of Bow Seals

AD-A043 448 NATIONAL ACADEMY OF SCIENCES-NATIONAL RESEARCH COUNCI--ETC F/G 20/4
PROCEEDINGS OF THE WORKSHOP ON NUMERICAL HYDRODYNAMICS.(U)
JUL 75 N00014-67-A-0244-0013

UNCLASSIFIED NL

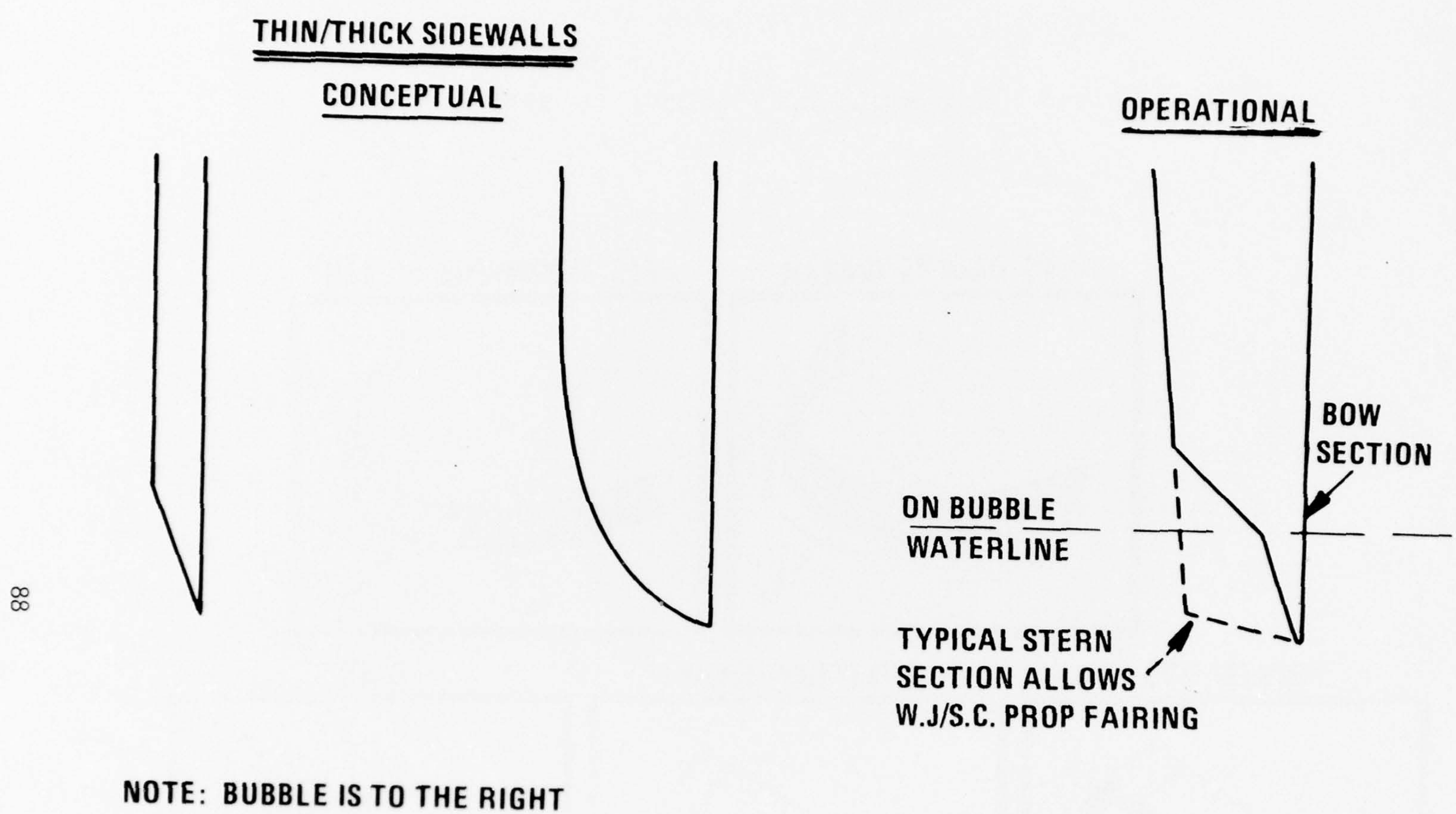
2 OF 5

AD A043 448

SIFIED

2 OF 5

AD
A043 448



CHARACTERISTIC	Thin (Conceptual)	Thick (Conceptual)	Operational
DRAG	VERY GOOD	POOR	GOOD
RESTORING FORCE	POOR	VERY GOOD	GOOD
CONTROL SURFACES/ STABILIZING APPENDAGES	NEEDS PITCH/ ROLL/YAW STAB.	CONFIGURATION DOMINANT (YAW STAB. ALSO REQ.)	GOOD MAY REQ. PASSIVE AUGMENTATION
PROPULSOR INTEGRATION WATERJET/S.C. PROP	POOR	GOOD	VERY GOOD

Figure 22. Comparison of Most Promising Sidewall Sections

the drag of each item sticking out from the basic streamlined body plus the drag of the streamlined body. Hoerner[16] has been the basic reference.

The seal drag is again treated empirically and the data is mostly proprietary. However, this does not preclude posing the general problem. Figure 19 illustrates the airflow and seal system for an SES. Possible seal functions are bubble retention, vertical positioning, turn control and contributing to pitch and heave stability. At the forward and aft ends of the craft we have a section like that illustrated in Figure 20. An inner seal may also be present, although no current proposals include one. The seal(s) may be of variable or constant stiffness and may vary from being a rubberized fabric to steel. The trade-off here is between a stiff seal which can develop restoring force for stability or a soft one which would have less drag but not contribute much to stability. It is not obvious whether the seal should be soft or stiff, but steel is an unlikely material. Generally a rubberized fabric with stays and pockets has been used at the bow, as illustrated in Figure 21. In plan form the bow seals are usually curved. The stern seals tend to be straight across in plan view and relatively stiff, especially in the SES. The skirts along the sides are dragged in a plane 90° from the skirts in front. Thus, the skirts away from the bow on a curved bow are dragged in a combination of the two ways.

The major design issue for an operational SES or ACV is seal life or wear resistance. One side of this issue is material selection or development, including the structural design of the seal to minimize its wear. The other side of this issue is the development of hydrodynamic loading models which can predict fatigue loading histories. There is little doubt that this is a hydroelastic problem.

The sidewalls of an SES contain the air cushion contribute to pitch and heave stability and possibly adversely to yaw stability, and house the propulsor and its drive train. Figure 22 compares three of

the most promising sidewall section shapes. The cushion is on the right and the shaped face of the sidewall is outboard. Drag is mostly frictional and is not a prediction problem.

3. Hydrofoil

Generally, available methods of drag prediction for fully submerged, subcavitating hydrofoils are adequate. The total drag can be separated as follows:

Foils
- friction
- wave
- induced
- form

Struts
- friction
- form
- spray

Foil-strut interference

Pod (if any)
- form
- friction

Form drag is usually not important unless a base vented strut is involved. Spray and interference drags are treated empirically. Friction drag is calculated based on wetted surface and Reynolds' Number. The most sophisticated treatment of induced drag is that due to Richardson.[17] It is a lifting surface theory including the effects of the free surface and Froude Number for fully submerged hydrofoils of no dihedral. Previously Windall[18] developed an unsteady theory. Neither accounts for struts or pods nor has either been thoroughly checked.

Besch and Rood[19] have begun checking both theories. Langan and Wang[20] have recently compared fifteen lifting surface computer programs with each other and with experiments. However, no free surface effects are included in these programs.

Cavitation drag, whether due to low local cavitation numbers on the foils or at intersections, e.g., pod-foil, is probably unimportant in magnitude. Prediction of cavitation inception is important from the structural damage viewpoint. Probably the only research areas for hydrofoils that are of principal future interest are: the effect of local waviness and roughness on cavitation and drag, separation prediction at a flap hinge, base vented strut drag and cavity formation, and the induced drag of biplane configurations. The latter problem will be of interest if we desire to design craft of much larger sizes. This is because the foils begin to weigh too much at the wing spans needed for good aspect ratios and because large wing spans impede practical operations if only a single wing is used.

Propulsors

The conventional propeller (either fixed pitch, controllable pitch or controllable/reversible pitch) is the natural propulsor for the SWATH ships because of their geometry and speed range. The other High Performance Ships can be treated as a group because their speed range results in considerable commonality in propulsors to be considered.

1. SWATH

High propulsive coefficients (P.C.'s) were expected and achieved for the SWATH.[21] A P.C. of 0.70 is typical. It would seem feasible to develop a method of analytically predicting the flow field around the afterbody with a view towards a totally analytical propeller design and optimization procedure.

2. ACV/SES/Hydrofoil

The candidate propulsors for the ACV, SES and Hydrofoil are:

- Subcavitating propeller
- Supercavitating propeller
- Semisubmerged supercavitating propeller
- Air propeller
- Water jet

In general the subcavitating propeller is not of interest. However, there have been recent proposals to build slow speed hydrofoils to take advantage of their seakeeping qualities. In this speed range, subcavitating propellers would be of interest. The main problem with subcavitating propellers is the prediction of the wake they see when behind foils and/or struts. Then knowing the wake, the prediction of cavitation and unsteady forces remains a problem to be solved.

The supercavitating propeller is beginning to approach the development status of the subcavitating propeller. A lifting line theory exists and has been programmed.[22] However, empirical corrections are needed and the problem becomes one of 'cut and try'. A lifting surface theory has been developed but has not yet been implemented on a computer. The semisubmerged supercavitating propeller is designed empirically.

Air propellers are principally of interest on ACV's because they allow the ACV to be amphibious. It would be possible to retract water propulsors in order to coast onto a beach but there would then be no positive control and no mobility over land. Because their diameter can be held down while their efficiency remains fully shrouded air propellers have been used primarily. Airborne noise remains a problem. Design problems center around the utilization of already developed hub mechanisms because of development costs. An adequate theory exists for the non-yawed condition. Yawed (separation) inflows are still an outstanding problem although Bell Aerosystems has done some work on the X-22 V/STOL aircraft.

Water jets have recently been utilized on PGH 2 (Tucumcari), the SES 100A, and PHM. Their efficiency is lower than a supercavitating propeller's but they are alleged to be more reliable because of the necessity of a complex mechanical transmission for the propeller. A good water jet design centers around the inlet part of the system. inlet must meet both take-off and cruise thrust requirements. This means that the mass flow at the lower take-off speed must be increased.

There are two basic types of inlet: ram and flush. The ram inlet (Figure 23 is the one on the SES 100A) results in an additional appendage with an added drag if this is the only use for the appendage. In the case of hydrofoils, a strut is there to tie the foils to the hull and so ram inlets are used. The flush inlet, Figure 24, is the more difficult design problem (this statement is not meant to minimize the difficulty of designing low drag, variable inlet area, ram inlets). This is because the flow (boundary layer) tends to separate as it tries to make the turn into the inlet. In an SES, this tendency is compounded by the very low sidehull submergences (a foot or two) which might result in ventilation or cavitation of the inlet lip or sideplates.

The current design procedure is to use a potential theory mathematical model, i.e., Johnson et al.[23] A physical model is then built and tested and, if necessary, the design is modified. Johnson et al[23] seem to have adequately addressed both internal and external cavitation and ventilation. However, the prediction of internal separation remains an important and unsolved problem. Pump design problems with adverse inflows and cavitation problems are common in water jet installations but are not addressed herein.

Seakeeping

SWATH

The conventional ship motions theory is reasonably good for SWATH geometries. However, it is becoming clear that a better treatment

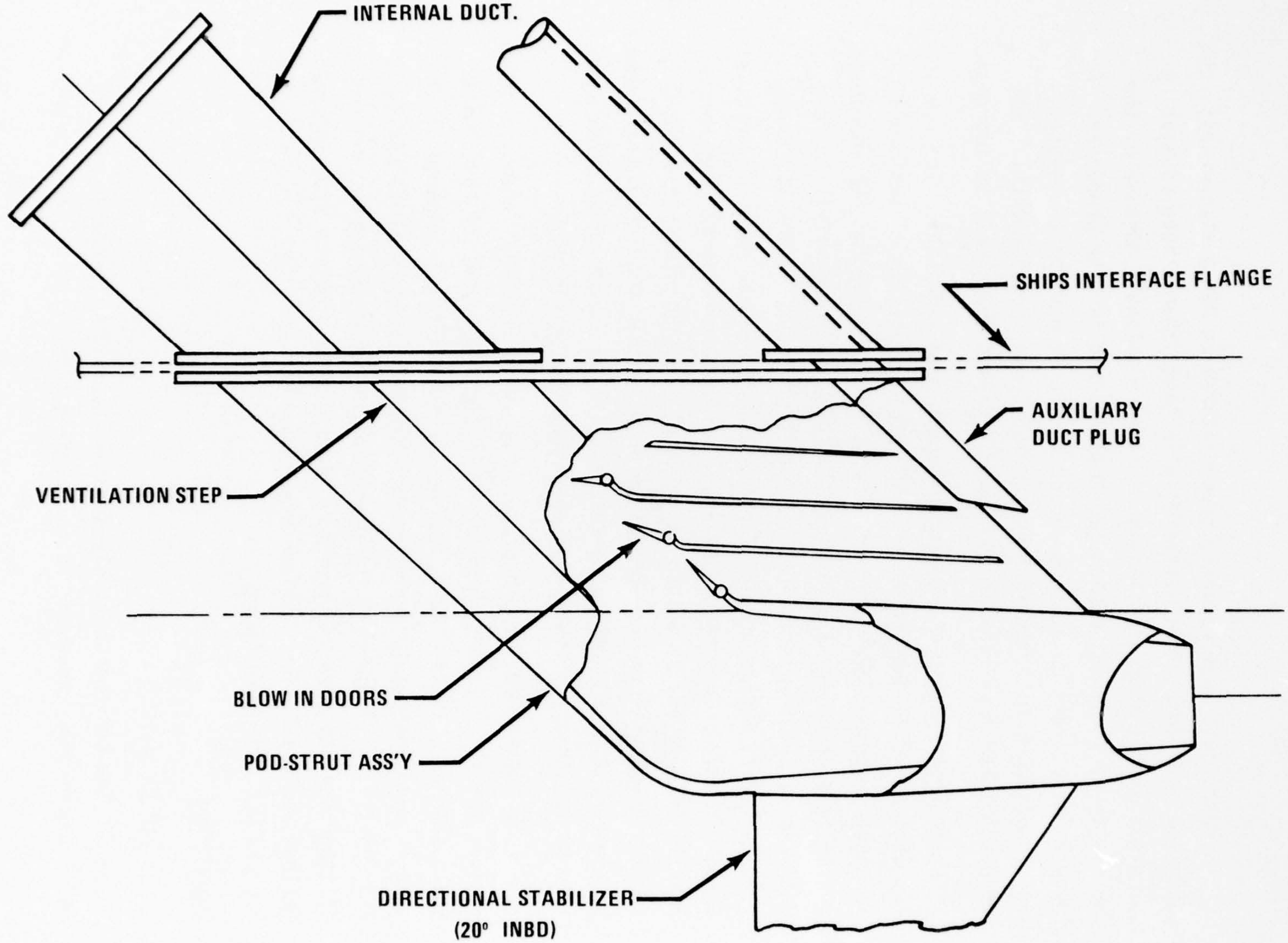


Figure 23. A Ram Inlet As Implemented on the SES 100A

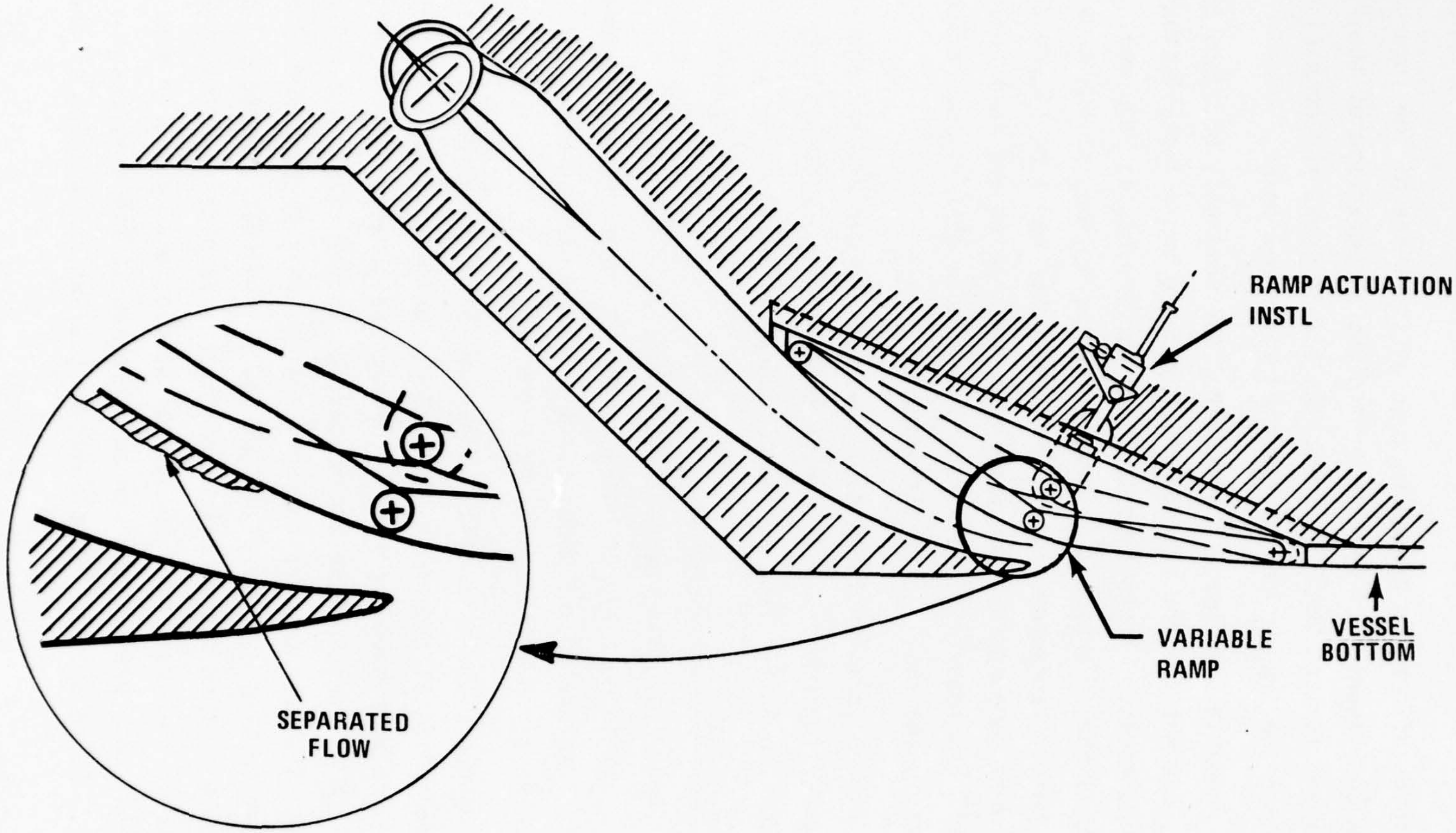


Figure 24. A Typical Flush Inlet. The Inset Illustrates Where Separation May Occur.

of viscous effects is needed because of the shape of the lower hulls. Figure 25 (Figure 7 of Reference 9) gives a comparison of theory and experiment for the SWATH II form. As can be seen, the comparison is poor near the theoretically predicted resonance point.

Probably some horizontal control surfaces will be added to SWATH hulls, but not for the reasons which require their installation on some Navy catamarans. Prediction of their performance is required. A potential theory approach to predicting the motions of the AGOR 16 (USNS HAYES) catamaran, not a SWATH, with her new fin, Figure 26, has shown very good correlation with both model tests and full scale trials. Hadler[24] has reported this work and concluded that viscous effects are apparently too small to be significant.

Figure 26 is included to illustrate how our elegant theories are frequently implemented. It can be seen that the fillet at the foil/hull intersection is far from optimum hydrodynamically!

Hydrofoil

The hydrofoil's seakeeping behavior is a function of its control system. It is possible to characterize its open loop performance adequately from theory and model tests and to design an excellent autopilot. Research in this area is not needed.

However, there is still an open problem in hydrofoil design, i.e., the prediction of the onset of flutter. While no cases of flutter or divergence have been reported by operational craft, a sound prediction technique is needed if we are to move to larger size craft. This is because as size increases the foil systems will become "more limber" in order to keep structural weight down. The best theoretical method available is that due to Besch and Lin.[25] It often predicts bending flutter in the wrong mode and so the predictions cannot be used. Its torsional mode predictions can be corrected to properly predict flutter speeds. Besch and Lin[26] are about to publish an updating of their work.

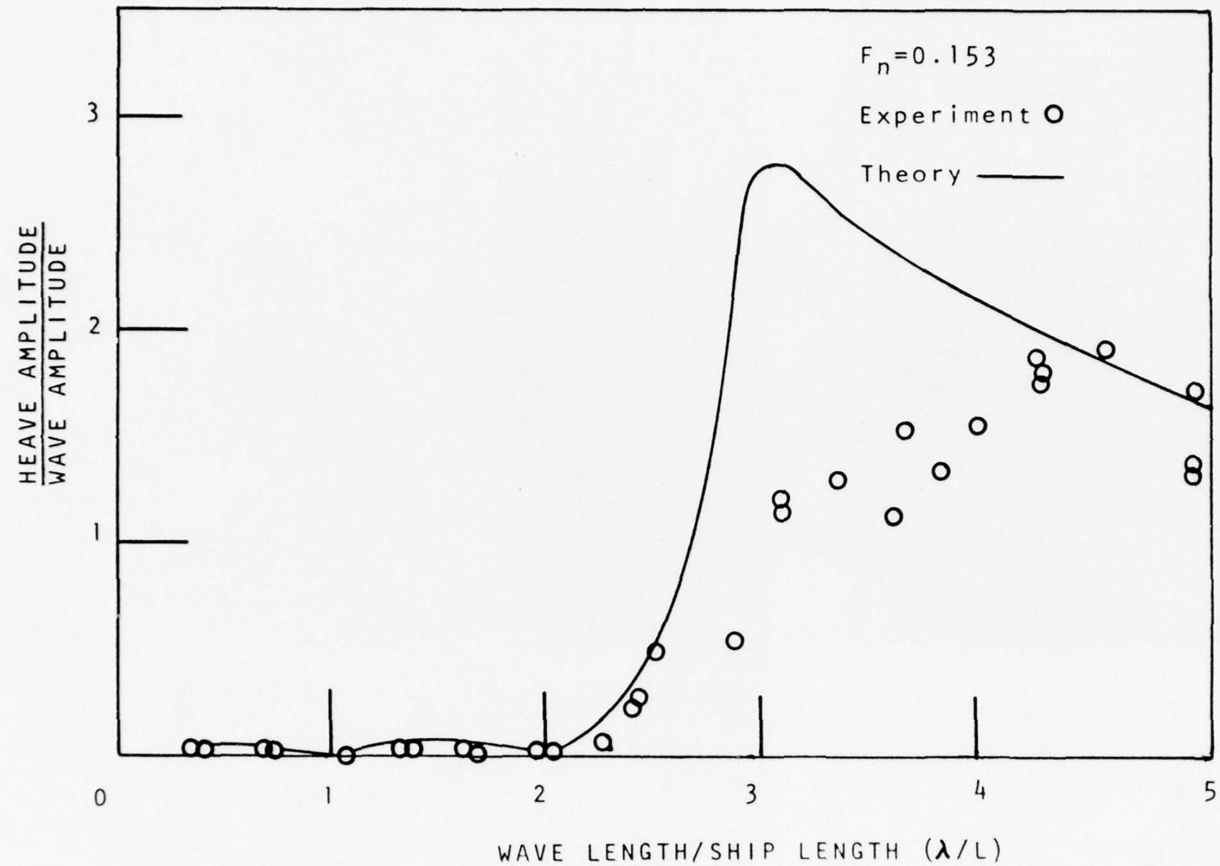


Figure 25. Comparison of SWATH II Heave Theory With Experiment

Figure 26a. The Anti-Pitch Foil of the USNS Hayes During Installation

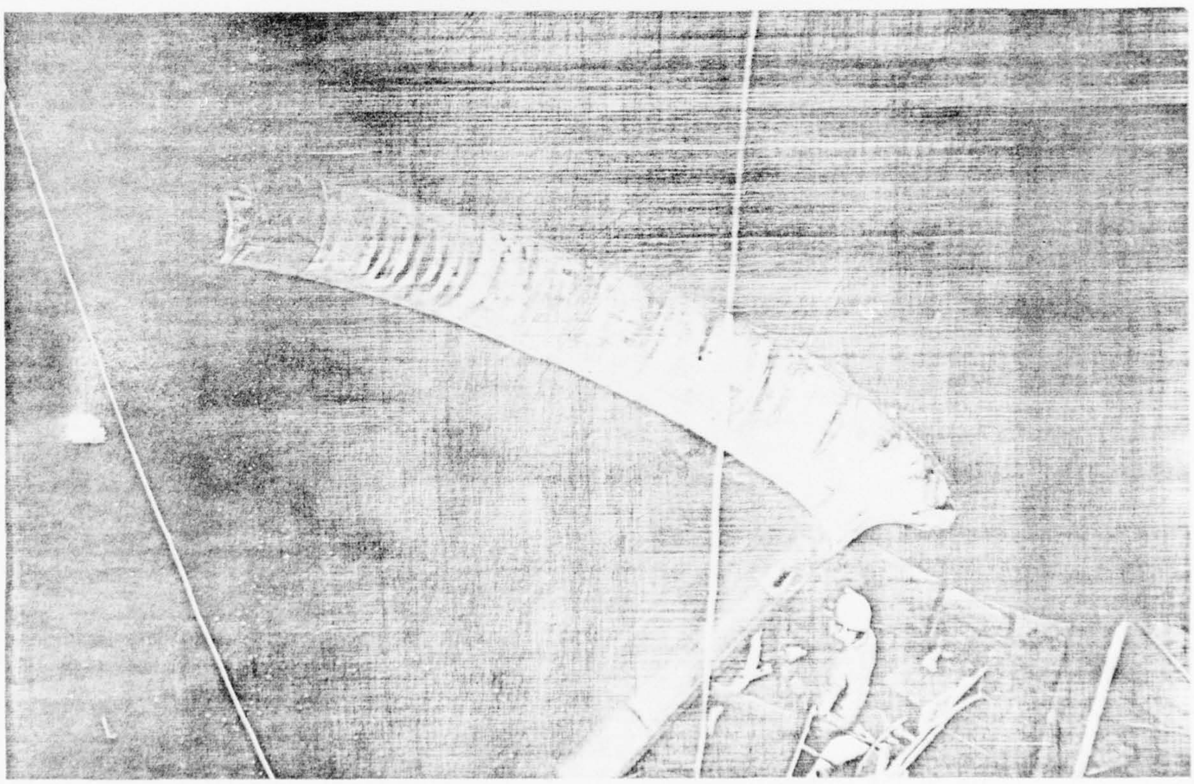


Figure 26b. The Fillet at the Hull Foil Intersection. Note the Poor Fairing at the Leading and Trailing Edges.

A related problem could occur if a biplane configuration was resorted to in a large size hydrofoil. If the upper foil with a flap is fairly close to the free surface, it could conceivably experience a synchronous oscillation with waves passing above it. That is, as the flap deflects to counteract the wave orbital velocities, it could force the foil into oscillation if the forces' frequency was near a natural frequency of the foil. Prediction of such vibrations would be complicated by the fact that a biplane configuration is a redundant structure.

SES/ACV

A completely rigorous general theory or model of the open loop behavior of SES's does not exist. The best one available is that developed by P. Kaplan, J. Bentson, and T. Sargent[27] for the Surface Effect Ship Project Office (SESPO). He models forces on pieces (e.g., sidewall forward seal) of the craft and then sums the forces to get the integrated force picture in the time domain. There is no reason to doubt this basic approach. However, it has not had a consistent, rigorous derivation. A great deal of work remains in modeling the hydrodynamics of the various pieces, Kaplan.[28] The following two paragraphs discuss two of the more important problems.

Modeling of each of the pieces is not thoroughly understood. For example, after an understanding of the steady state response of the seals is acquired one must then look at their dynamic response. In principle, it should be possible to model the seals' responses and the forces and moments that they exert on the ship by using a sufficiently fine-meshed finite element representation for them in either a steady or unsteady mode. It has been found by SESPO that the seal motions are of a much higher frequency than the ship motions. The consequence of this is that the computational time to do SES motions calculations is quite high if the seals are directly modeled. What is needed is an

approximate model for the seals which represents their macroscopic, but not their microscopic, behavior. In general, a hydrodynamics of flexible surfaces moving at high speed on the ocean's surface is needed.

The dynamic forces on sidewalls are also not well understood. They are generally extremely low aspect ratio, longitudinally curved planing surfaces. Prediction of their lift and induced drag is empirical, as is the prediction of their cross flow drag. The cross flow drag prediction is complicated by the fact that the water level under the bubble is lower than the water level outside the craft. This is a function of speed, where near hump there is a considerable difference in level and at high speed there is almost no difference.

The ACV represents a far easier prediction problem because the complexity of the sidewalls is eliminated.

Control

Of particular interest in theoretical models of the dynamics of these craft is the ability to model failure modes with a view to predicting catastrophic failure. These situations are beyond "small perturbation." If these models are adequate to predict the vehicles' overall response, they can then form a sound basis for structural load predictions. While they may not be adequate for analyzing local situations, like the slamming of a panel, they certainly would properly prescribe the overall velocity and angle of entrance picture. This should then be a sufficiently global basis for building a local hydroelastic model.

Special Problems

There are various special types of problems that could not be conveniently placed in the above categories. They are listed and briefly described below.

The prediction of the airflow over aircraft landing areas is a significant problem. This must include the effect of gas turbine exhaust plumes.

The prediction of tip vortex paths is of interest. They could be coming from control skegs on an SES or from forward hydrofoils. The problem is to keep them from impinging on propulsors or other control surfaces.

The modeling of the airflow within an SES or ACV bubble, including the prediction of pressure gradients due to waves and the effect of large percentages of spray, is desirable.

The prediction of fan performance when operating against fluctuating pressures of various frequencies is needed to predict the dynamics of ACV's and SES's in seaways.

Summary

In general the hydrodynamic problems to be addressed for high performance ships involve more major issues than for conventional surface ships. However, the level of detail of the hydrodynamics may have to be just as sophisticated.

The best understood vehicle is the hydrofoil; the poorest understood is the SES. The weaknesses in the hydrofoil are mainly in extent of data bases and in detail considerations. The major weakness of the SES area is in modeling its dynamics. The SWATH area is also weakest in notions' predictions.

ACKNOWLEDGEMENTS

The authors are indebted to many engineers and scientists from the Naval Ship Research and Development Center, the Naval Ship Engineering Center and Hydronautics, Inc. for their suggestions and help. Special thanks go to:

Donald N. McCallum
David P. Lewis
John J. Slager
Roderick Barr

Without the assistance of these individuals, this paper could not have been prepared.

REFERENCES

1. Lackenby, H., "B.S.R.A. Resistance Experiments on the "Lucy Ashton", Part III -- The Ship-Model Correlation for the Shaft-Appendage Conditions," RINA Volume 97, 1955.

2. Todd, F.H., "The Model-Ship Correlation Program," Marine Technology, Volume 3, Number 2, 1966.

3. Kowalski, T., "Boundary Layer Suction Influence on Scale Effects of Self-Propelled Models," RINA Volume 107, 1965.

4. Burcher, R.K., "Developments in Ship Maneuverability," RINA Volume 113, 1971.

5. Hoult, Fay, and Forney, "A Theory of Plume Rise Compared with Field Observations," M.I.T. Fluid Mechanics Laboratory Publication Number 68-2, March 1968.

6. Pien, P.C. and C.M. Lee, "Motion and Resistance of a Low Waterplane Area Catamaran," Ninth Symposium on Naval Hydrodynamics, Paris, France, August 1972.

7. Chapman, R.B., "Hydrodynamic Drag of Semi-Submerged Ships," ASME Paper 72-WAOCT-5, November 1972.

8. Lin, W.C. and W.G. Day, "The Still Water Resistance and Powering Characteristics of SWATH Ships," Paper Number 74-AIAA/SNAME, Advanced Marine Vehicles Conference, San Diego, February 1974.

9. Hawkins, S.and T. Sarchin, "The Small Waterplane Area Twin Hull (SWATH) Program - A Status Report," Paper Number 74-324, AIAA/SNAME, Advanced Marine Vehicles Conference, San Diego, February 1974.

10. Murray, L.O., "Predictions of Resistance for a Single Hull of A Small Waterplane Area Twin Hull (SWATH) Form Represented by Models 5301 and 5301-1," NSRDC Evaluation Report 396-H-12, September 1973.

11. Chapman, R.B., "Spray Drag of Surface Piercing Struts," Paper Number 72-605, AIAA/SNAME/USN Advanced Marine Vehicles Meeting, July 1972.

12. Sarchin, T.H., P.A. Chatterton and C.G. Kennell, "Small Waterplane Area Twin Hull (SWATH) Combatant Ships Feasibility Design Studies," Volume 1, NAVSEC Report Number 6114-74-8, January 1974.

13. Newman, J.N. and F.A.P. Poole, "Wave Resistance of a Moving Pressure Distribution in a Canal," DTMB Report 1619, March 1962.

14. King, N., P.T. Klimes and J. Sladky, "Principles of Surface Effect Vehicles," Notes for G.W.U. Course Number 60, June 1973.

15. Chaplin, H.R. and A.G. Ford, "Some Design Principles of Ground Effect Machines," Section D - Drag, DTMB Report 2121D, June 1966.

16. Hoerner, S., "Aerodynamic Drag," 1951.

17. Richardson, J.R., "Pressure Distribution on a Hydrofoil of Finite Span Near the Free Surface," Report Number 53/3, Engineering Research Association, Toronto, Canada, June 1966.

18. Widnall, S.E., "Unsteady Loads on Hydrofoils Including Free Surface Effects and Cavitation," M.I.T. Fluid Dynamics Laboratory Report Number 64-2, June 1964.

19. Besch, P.K. and E.P. Rood, "Evaluation of the Lift and Moment Predictions of the ERA Computer Program for a Hydrofoil Beneath the Free Surface," NSRDC Technical Note 138, June 1969.

20. Langan, T.J. and H.T. Wang, "Evaluation of Lifting-Surface Programs for Computing the Pressure Distribution on Planar Foils in Steady Motion," NSRDC Report 4021, May 1973.

21. Lin, A.G.M., L.B. Crook and L.O. Murray, "Prediction of Resistance and Propulsion Characteristics for a Small Waterplane Area Twin Hull (SWATH) Form Represented by Model 5287," NSRDC Evaluation Report 396-H-08, December 1972.

22. Yim, Boh-yum, "A Preliminary Design Method for Supercavitating Propellers," NSRDC Report 4265, unpublished.

23. Johnson, V.E. Jr., R.J. Etter, P. VanDyke, G.M. Poquette, J.R. Hill, L.K. Stephens and N.R. Stark, "Design and Performance of Diffusers, Fixed-Area Inlets and Variable-Area Integrated Inlet-Diffusers, Sub-Systems," Technical Report Number 7152-1, August 1972.

24. Hadler, J.B., C.M. Lee, H.D. Jones and J.T. Birmingham, "Ocean Catamaran Seakeeping Design Based Upon the Experiences of USNS HAYES," Transactions SNAME 1974, to be published.

25. Besch, P.K. and Y-N. Lin, "Bending Flutter and Torsional Flutter of Flexible Hydrofoil Struts," NSRDC Report 4012, February 1973.

26. Besch, P.K. and Y-N. Lin, "Hydroelastic Design of Subcavitating and Cavitating Hydrofoil Strut Sections," NSRDC Report 4257, 1974, unpublished.

27. Kaplan, P., J. Bentson and T. Sargent, "Advanced Loads and Motions Studies for Surface Effect Ships (SES) Craft, Part I - General Development of Modified Mathematical Model," Oceanics Inc. Report 73-98A, June 1973.

28. Kaplan, P., "On Experimental Study of SES Craft Lateral Hydrodynamic Forces and Moments," Oceanics Inc. Report 73-97, May 1973.

APPENDIX A

DESIGN PHASES FOR CONVENTIONAL SHIPS AND THEIR RELATION TO THE HYDRODYNAMIC DESIGN DEVELOPMENT

Currently, five design phases are identified for conventional surface ships:

- Feasibility Studies - Emphasizing cost vs performance trade-offs
- Conceptual Design - Emphasizing absolute sizing of the ship
- Preliminary Design - Emphasizing ship system optimization and definition
- Contract Design - Emphasizing development of a procurement specification
- Detail Design - Emphasizing development of working drawings

For each design phase, the objectives and level of design detail addressed is described in the following paragraphs. Interfaces with hydrodynamic design development are discussed.

1. Feasibility Study Phase

Feasibility studies are performed to establish the cost vs performance trade-offs for a new design. Performance is expressed in terms of speed, endurance, major payload items and other special features, e.g., side protection systems or special shops. A primary objective is to identify the trade-offs in a self-consistent manner and relatively correctly quantitatively. It is not unusual to generate more than 200 ship design studies in this phase of design. Each study will reflect one set of performance requirements and each will represent the "best" (say least acquisition cost) of perhaps 20 to 30 alternate configurations

which all meet the performance requirements. Thus, a total of 4,000 to 6,000 design studies may be developed.

In the feasibility study phase, computers are frequently utilized to develop alternative ship design studies. Whether the studies were performed by hand or computer, the various aspects of each design are treated at a rather gross level of detail. Design criteria invoked and analysis procedures utilized are generally simplified but, none-the-less, very important due to their influence on predicted ship cost and performance and hence, on high level Navy decisions on the requirements for the new ship.

The naval architect/hydrodynamicist contributes to this phase of design in three important ways. The first is by placing constraints on the range of variations which are studied. For example, maximum installed shaft horsepower per shaft must not exceed some value. The second is by providing criteria which the design studies developed must satisfy. An example would be the required freeboard as a function of length based on seakeeping (deck wetness) considerations. The third is by predicting, for each study developed, those key hydrodynamic performance characteristics which are selected for evaluation. In all cases the calm water speed/power characteristics of alternative studies are predicted and, in some cases, other aspects of performance, notably seakeeping characteristics, are assessed.

2. <u>Conceptual Design Phase</u>

The objective of the Conceptual Design Phase is to develop the selected feasibility study in sufficient detail to:

a. define the Top Level Requirements for the new design,

b. obtain an approximately correct end cost estimate

c. identify and resolve all major technical risks associated with the design.

A firm basis for initiating Preliminary Design must be provided. Overall ship geometry and weight are established. Type of propulsion machinery is selected along with ship speed and endurance. Platform subsystem trade-offs are not done, only standard approaches are assumed. Usually one, and sometimes two variants are carried through this phase of design.

Due to the nature of the Conceptual Design Phase, the products of that phase include a Conceptual Baseline Design which satisfies the Top Level Requirements for the new ship. Thus the two subsequent steps in the design process can both be considered to have as their objective the optimization, as well as the detailing, of the selected Conceptual Baseline Design. The detailing/optimization process is divided into two phases: Preliminary and Contract Design.

3. Preliminary Design Phase

Preliminary Design results in a complete and self-consistent engineering definition of the ship. By this time, only one ship will be under consideration. Thorough platform subsystem trade-offs are defined. All subsystems are sized based on the demands of the other ship systems that they must support. As a result of the platform subsystem trade-offs and consequent concept selections, the ship's size and weight may change. This will result in a changed end cost estimate. Ideally, there should be no change in ship size once Preliminary Design is complete.

The hull form characteristics which are established in the Preliminary Design phase are those which have the dominant influence on hydrodynamic performance: the principal hull dimensions (length, beam, draft and depth) and form coefficients (prismatic and maximum section coefficients) as well as the longitudinal distributions of waterplane area and immersed hull volume (bulb and transom characteristics along with LCB and LCF positions). The selection of these characteristics is based primarily on resistance, propulsion and seakeeping considerations

traded-off against cost, internal arrangement constraints and other non-hydrodynamic factors. A preliminary body plan serves as the basis for studies to determine the need for motion stabilization and, if required, to select the type and size or capacity of the motion stabilization system. At no time are studies to optimize hydrodynamic performance conducted which are divorced from the realities of the developing ship design. Thus the general approach which is used is to develop a set of realistic alternatives, evaluate each for cost, performance, and other aspects, and finally, based upon some established selection criterion, choose the "optimum" solution.

During the Preliminary Design Phase, model testing is generally no longer done. This is because of the greatly increased costs of model testing; the desire to explore more alternatives than one has time to model and test; and, lastly, because improved analytical prediction techniques have been developed in many areas. Thus predictions of hydrodynamic performance at the end of the Preliminary Design Phase are generally made using analytical methods.

4. Contract Design Phase

The objective of Contract Design is to define, in a sound contractural document, the ship to be built. This is done primarily by the preparation of a Ship Specification and, secondarily, by the preparation of contract drawings and contract guidance drawings. Further detailing of the ship's subsystems takes place during this phase, especially with respect to the drawings. Considerable effort is expended to avoid contradictions within the Ship Specifications and to insure its completeness. Changes do occur to the ship design as problems are identified and solved but the changes are usually minor and do not affect the ship as a whole.

During the Contract Design Phase, the hull form and appendage configuration details (including propulsor geometry and hull interfaces)

are finalized along with the specifics of the geometry of any motion stabilization system incorporated in the ship design. Model testing is utilized to a great extent, as an aid in both developing design details and making the final Contract Design predictions of hydrodynamic performance. The final ship lines drawing is invariably a contract drawing from which the shipbuilder may not deviate.

5. Detail Design Phase

During this phase, the Ship Specification and drawings developed during the Contract Design Phase are translated into working drawings and numerical control tapes to permit ship construction to begin. Procurement specifications for Contractor Furnished Equipment and Material are developed. Additional hydrodynamic studies are only performed during this design phase if some unusual problem develops.

DISCUSSION

T. F. Ogilvie

I must say that I was a little appalled to find that one of the accomplishments mentioned draws upon the work of Newman and Poole which dates from, as I recall, 1961-62, and that you are using the results of rectangular pressure distributions for other kinds of vehicle shapes. These results are simply not going to be correct. It is disappointing that with nearly 13 years to do analyses we have not made more progress in this area, and it makes me wonder what we are doing here today.

P. A. Gale

I can appreciate your concern, and I would ask Mr. Buck if he would like to respond to your specific comments.

J. Buck

Yes. I tend to agree with you about the SES, but I don't want to leave the impression that we depended solely on Newman and Poole; other techniques were incorporated. We have a method for predicting wave making resistance of a pressure field moving along a surface which is quite interesting and gives us good results. I was using a different geometric pattern and footprint in the water besides the rectangular pattern. This allows us to consider a rigid sidewall rather than a fully skirted one, and gives an indication of how the seals interact with that wave-making drag.

While I have the floor let me also respond to the speculation as to our purpose here today. The answer, I believe, goes back to my earlier comment about the need to establish priorities. Represented here today are researchers interested in whether specific problems are limited by physics, hydrodynamics or numerical analysis; laboratory people who gather data and do basic model tests; and, designers interested in tools and

materials. With this kind of representation we should be able to determine those things that need to be done, and their priority position on the resulting list.

At present we are very high on that learning curve which deals with the design of conventional ships. Therefore, a concentration of funds and efforts in this area will return very little. Just the opposite is true with respect to our position on that learning curve which relates to the design of nonconventional, high performance ships and craft. Therefore, I believe that a healthy dialogue here today can be quite useful in setting those priorities which are most likely to achieve, within available resources, the kinds of ships we will need in the 1980's and 1990's.

P. Lieber

It seems to me that the preceding remarks go far towards indicating a rather pressing need for intensifying the interplay between the various specializations represented here today. By this interplay I think we can develop a capability which is both complimentary and mutually reinforcing. By way of a specific proposal, I would suggest that we spend more time, on a day-to-day basis, with laboratory situations.

W. Webster

I wonder if I could get some clarification on an apparent difference between the last two papers. One states that we don't have enough information to determine changes in resistance, and the other states that resistance is no longer a problem.

P. A. Gale

In one sense, both positions are correct. Traditionally, a naval architect would simply draw up a set of lines for a new ship and that would be sufficient for model testing, building and the like. More recently we want to explore alternatives, and to examine the pros and cons of competing designs. For instance, the optimum from a resistance standpoint may not be optimum from the standpoint of seakeeping. etc. The

position taken by Dr. Cummins was that once you have a hull form the model basins can arrive at acceptable predictions regarding resistance and other performance characteristics. My statement was that, at an earlier design stage, we need the tools that will allow us to examine alternatives and tradeoffs. Twenty years ago we simply built a model for each alternative and tested it. This we no longer do because of prohibitive costs.

M. Tulin

As a designer, would you like to have numerical based methods for determining wave resistance of a ship with more or less arbitrary lines?

P. A. Gale

Absolutely. Now, in a few minutes of computer time, we can produce printouts of several alternate sets of lines. For these, particularly in the case of head seas, we can predict seakeeping characteristics, and compare relative motions. But, in such areas as resistance and propulsion we are right back to Taylor. We definitely need the capability to do tradeoff studies in these areas.

W. E. Cummins

I would like to call attention to the fact that, in dealing with small changes in alternative designs, Taylor's Standard Series is not the only tool available to us. In addition, we have available the log of every model test that has been run since 1898. And, some of the early tests were very good. So, when ship engineers throw a design at us and want to know what it is really like, we do a computer search, pick out two or three designs related to the one they are interested in, and tell them how to improve it. This process can tell us what to do under certain constraints to achieve a 10 to 15 per cent improvement. Therefore, as long as we are in the domain of experience we can provide guidance to the designer on such matters as resistance, powering, and the sensitivity of powering to hull changes.

Let me also speak to the earlier comment about the lack of work on the SES, and the resulting lack of guidance to the designer. Those of us who have pushed long and hard for more attention to this area will agree that this is an appropriate area to question. But, the answer is quite simple. In the Navy today there is very little money available for such basic research. True, a great deal of money is going into the design and construction of the 2,000 ton SES. But none of that money is allowed to be used for basic research and, they are watching expenditures very closely.

As a result of this funding shortage the technology we are concerned with here is virtually untouched. Exceptions are a little work supported by NAVSHIPS, and one project out of ONR.

J. Breslin

I believe that it is worth pointing out that when a naval architect looks for a day-to-day tool by which to calculate the resistance of a ship to waves, or to do tradeoff studies such as mentioned earlier, he is looking for an accuracy which is phenomenal. A PC prediction, for instance, is rather phenomenal relative to the ability of linear theory or other theory to account for second order effects. This is particularly true in the case of conventional ship design where the expected performance and the norms of competing hulls, etc., have been under development since the time of Noah. So, we have to divide these problems into two broad categories -- those in which we want to look for engineering tools having a high degree of accuracy suitable to tradeoff studies, and those out on the frontier of the field such as surface effect ship mechanics.

Another point about which I am a little sensitive is that of vibratory forces produced by propellers. There have been other methods advanced over the years so I don't want to downgrade Vorus, but there must have been some arithmetic mistake in the comparison of Vorus' theory and the vertical force in your example. We should be able to achieve better results today on the back of an envelope.

SESSION II

NUMERICAL HYDRODYNAMICS

NUMERICAL HYDRODYNAMICS AS A MATHEMATICAL SCIENCE

Garrett Birkhoff
Department of Mathematics
Harvard University
Cambridge, Massachusetts

INTRODUCTION

1. Analytical Fluid Dynamics. The idea of considering fluid dynamics as a purely mathematical science was clearly envisaged by Lagrange in 1788, when he wrote [1, p.3]:

> "One owes to Euler the first general formulas for fluid motion....presented in the simple and luminous notation of partial differences..... By this discovery, all fluid mechanics was reduced to a single point of analysis, and if the equations involved were integrable, one could determine completely, in all cases, the motion of a fluid moved by any forces....."

Lagrange was, of course, basing his statement on earlier thinking and speculations of Newton, D. Bernoulli, d'Alembert, Euler, and others. See Hunter Rouse and Simon Ince, "History of Hydraulics," Iowa Inst. Hydraulics, 1957, especially Chapters VII-VIII.

The partial differential equations referred to by Lagrange were those for potential flow (see §6); he was unaware of the basic phenomena of vorticity, viscosity, shock waves, and turbulence. But by the end of the 19th century, various other mathematical models of fluid flow had been constructed and studied by many mathematicians and mathematically minded scientists like Stokes, Kelvin, Helmholtz and Rayleigh. Their many ingenious results are masterfully summarized in Lamb's encyclopaedic treatise "Hydrodynamics" [24], whose first edition

PRECEDING PAGE BLANK

appeared in 1879 and whose sixth edition came out in 1932. Half of Lamb's treatise is devoted to potential flows, whose theory still constitutes the best developed branch of analytical fluid mechanics.

However, more recent books on hydrodynamics have tended to downgrade the importance of potential flows, and indeed to discredit Lagrange's vision. Thus in Goldstein's influential "Modern Developments in Fluid Dynamics,"* partial differential equations do not appear at all until the third chapter, and the treatment becomes increasingly more empirical as the treatise progresses. It is hard to believe that Lamb was originally to be the general editor.

And indeed, Lagrange's concept of fluid dynamics as a deductive science has become badly fragmented. There exist today at least five very distinct analytical formulations of the phenomena of fluid mechanics, whose differential equations (found in Lamb) must be integrated if we are to understand these phenomena. I have listed the relevant models in Table 1 for convenient reference, together with their originators and the corresponding chapters of Lamb's book. To make numerical hydrodynamics into a fully developed science, one should be able to integrate them all numerically with the aid of high-speed computers!

2. Numerical Fluid Dynamics: During the century 1845-1945, Lagrange's concept of fluid mechanics as a mathematical science gradually lost scientific credibility. To apply analytical methods to real problems in hydraulics and aerodynamics, engineers had to make allowances for many factors that pure theory ignored. Thus they had to compute skin friction and viscous wake drag arising from flow separation, as well as turbulence effects, largely on the basis of empirical formulas.

*Two volumes, Oxford University Press, 1938.

TABLE 1. FIVE STANDARD ANALYTICAL MODELS FOR FLUID DYNAMICS

Originators	Lamb Chapters	Remarks
Euler-Lagrange Potential Flow	III-VI VIII-IX	Solid Boundaries, vs. Free Boundaries Interfaces, Slipstreams Gravity Waves
Helmholtz-Kelvin Vorticity	VII	Cyclones and Anticyclones
Navier-Stokes Stream Function Incompressible	XI (pp. 563-695)	Asymptotic: Boundary layer, Lubrication
Helmholtz-Rayleigh Acoustic Waves Wave Equation	X	Signals, Wave Fronts Analogy with Geometrical Optics and Rays

Comment: The remaining models are less well developed: Their mathematical theory is not rigorous, and I will not discuss them. In any case: The optimal numerical technique depends on the mathematical model.

Rankine-Hugoniot Supersonic Flow and Shock Waves	X, Cont.	Bulk Viscosity?
Prandtl-Taylor (Reynolds) Turbulence Statistical	XI' (pp. 663-96)	$u(\underline{x},t;\omega)$

Model tests in towing tanks, wind tunnels and water tunnels became increasingly the basis of practical design work, suitably corrected for "scale factors".

This is not to say that the Euler-Lagrange potential flow model was discarded. It continued to give great insight into the behavior of ocean waves and tides, the wave resistance of ships, ship motion in a seaway, cavitation in high-speed flow, and countless hydraulic phenomena -- not to mention (for other reasons) the seepage of groundwater and petroleum.

However, potential flow became thought of as an approximation to the more exact Navier-Stokes equations, somewhat like Prandtl's (laminar) boundary layer equations. To be sure, there was much evidence for the fundamental correctness of the Navier-Stokes equations in the realm of low speed (< 200 m.p.h.) flow. Especially convincing were the classic experiments of Stanton and Pannell, which showed that the critical Reynolds number was the same in air as in water, even though their molecular structure is very different (gas vs. liquid). But exact analytical solutions were so hard to come by that fluid mechanics came to be thought of as a complex mixture of theory and experiment, in which mathematics by itself should not be taken too seriously.

It was therefore an electrifying experience for me and others to hear von Neumann propose in 1945, at the First Canadian National Mathematical Congress, that computers could be used instead of wind tunnels to integrate the partial differential equations of fluid mechanics. It was a reaffirmation of Lagrange's Thesis, that fluid mechanics could be treated as a mathematical science, and it is with this thesis that I shall be concerned today -- in the context of naval problems.

Indeed, I will ask you to distinguish three different kinds of mathematical models: (a) philosophical models, (b) engineering models, and (c) scientific models. By a philosophical model, I mean a model which gives philosophical insight into qualitative features of reality. By an engineering model, I mean one which conforms quantitatively to reality with sufficient accuracy for engineering design purposes (often strength up to ±50%, with a factor of two for safety, in structural design). By a scientific model (or theory), I mean one which can be used to predict phenomena under controlled laboratory conditions with great precision, the intention being to eliminate or understand all deviations, modifying the theory if necessary.

The distinction between these is well illustrated by the mathematical model of a bore, invented by Rayleigh and developed by Stoker and others.* Long waves of infinitesimal amplitude in shallow water behave like compression waves in a polytropic gas with $\gamma = 2$ [1, §15], provided that the velocity is nearly independent of depth. This analogy enables one to explain philosophically the formation of bores, and is also of potential engineering value as a way of modelling approximately the relation between the depth of water on the two sides of the bore and its forward velocity. However, it is unscientific: real bores have much turbulence, and a velocity which presumably is much less on the bottom than at the surface, and the equation of energy conservation in a perfect gas with $\gamma = 2$ certainly does not apply exactly.

3. Von Neumann's Legacy, I: Von Neumann had only 10 years in which to develop his ideas. The boldness of his imagination becomes

*J.J. Stoker, "Water Waves", Interscience, 1957, §10.6; R. Courant and K. Friedrichs, "Supersonic Flow and Shock Waves," Interscience, 1948, §62. For engineering models, see Th. von Kármán, Bull. Am. Math. Soc. 46 (1940), 615-83, and Atti Int. Congr. Mat. Bologna (1928), vol. 1, pp. 301-14.

especially impressive when we consider the limited capabilities of the leading digital computers of the time: the ENIAC with its negligible "memory" and Aiken's electromechanical Selective Sequence Calculator. Unfortunately, his Montreal talk was not recorded and never written up; I had the unenviable task of speaking after him on "Universal Algebra", a topic which I considered imaginative enough but which paled into insignificance when given just after von Neumann spoke!*

Today, nearly 30 years after his Montreal talk and two decades after his untimely death, it seems timely to review von Neumann's papers and reports on numerical hydrodynamics more closely, and to reevaluate his vast legacy of ideas with the wisdom of hindsight. In making this review, we should remember that his ideas were engendered during World War II (against Hitlerism) and the Cold War (against Stalinism) that followed it almost immediately. In those years, the highest priority problems were the design of atomic (U-235 or plutonium) and hydrogen bombs.

Taylor Instability: One of von Neumann's major efforts in numerical hydrodynamics concerned so-called Taylor instability: the instability of the interface between two homogeneous incompressible, non-viscous fluids of different densities when accelerated from the lighter towards the heavier fluids. Besides being important in atomic bomb design, Taylor instability was known to be an important factor in destroying the symmetry during the collapse phase of (nearly) spherical

*See Proc. First Canadian Math. Congress, Univ. of Toronto Press, 1946, pp. 310-26 and p. xii. The next talk (pp. 327-37) by Hartree on the solution of partial differential equations by the differential analyzer, which was the leading computer at the time (it was an analog computer!).

underwater explosion bubbles,* a subject of perennial interest to the Navy. Von Neumann's proposal for computing this stability numerically was presented in edited form in a joint paper with A. Blair, N. Metropolis, A.H. Taub, and M. Tsingou.† This paper evolved from his earlier thinking, embodied in Los Alamos Report LA-2165; see also the brief reports contained in Selections 31 and 32 of [33, Vol. VI], dating back to 1953.

I feel well qualified to judge this work, because I was also working part time at Los Alamos on Taylor instability during the years 1953-59, in collaboration with R. Ingraham and David Carter. Indeed, I already have reviewed both Taylor and Helmholtz instability in [35, pp. 55-77], and discussed there the difficulty of predicting their evolution accurately.

My judgement is that both von Neumann's and my calculations‡ can best be summarized as pioneer efforts, whose interest today is primarily historical;‡ cf. also §9 and §16 below.

<u>Numerical Weather Prediction</u>: Von Neumann was also fascinated by the possibility of basing weather forecasting on differential equations describing the motion of the atmosphere, with the help of a computer.‡‡ In more exuberant moments, he even mentioned the enormous "payoffs" which might come from weather control, and his ideas have stimulated an enormous activity whose current status will be summarized by Dr. Leith.**

*See [7, Ch. XI, §13] and Robert H. Cole, "Underwater Explosions", Princeton Univ. Press, 1948, Sec. 8.7.

†Math. Tables Aids Comp. 13 (1959), 145-84; [24, Vol. V, pp. 611-52].

‡Mine were summarized in Los Alamos Reports LA-1862 (1954) and LA-1927 (1956).

‡‡For earlier ideas on the subject, see Lewis F. Richardson, "Weather Prediction by Numerical Process," Cambridge University Press, 1922.

** Hopefully, Charney's forthcoming Von Neumann Lecture, to be published in the SIAM Review, will contain even more pertinent remarks about von Neumann's pioneer work and influence.

The only trace of von Neumann's fascination with these possibilities which I have been able to find in his Collected Works consists of a single paper [33, Vol. VI, #30] with Charney and Fjörtoft on "Numerical integration of the barotropic vorticity equations". It is very clear that he regarded the calculations discussed in this paper as only "an essential first step in the general program", and that he felt greatly limited by the small size and speed of the computers then available. His solution on the ENIAC of the crude approximation which he used in these calculations must, indeed, be regarded as a brilliant achievement in both theoretical hydrodynamics and computer science!

Since my reference to this work is primarily for purposes of orientation, I shall only make a few remarks about it. First, the model used was "the quasi-geostrophic, non-divergent vorticity equation, in which the sole dependent variable is the height z of a fixed isobaric surface".* Thus the authors considered the function $z(\phi,\theta,t)$, where ϕ is latitude and θ is longitude. Using the vorticity

$$(3.1) \qquad \eta = \frac{g}{f}\nabla_s^2 z + f \quad (f = \text{Coriolis parameter}),$$

where ∇_s^2 is the surface spherical Laplace operator. To determine the (horizontal) velocity field $\underline{v}(\phi,\theta,t)$, as it does in an incompressible fluid, they used the approximate integrated vorticity equation†

$$(3.2) \qquad \partial\bar{\xi} = -\bar{v}\cdot\nabla(K\bar{\zeta} + f),$$

where K is a dimensionless near-constant, and

$$(3.3) \qquad \partial(\nabla_s^2 z)/\partial t = J_s(\eta,z), \quad J_s = \text{spherical Jacobian}.$$

*Specifically, the 500 mb level.

†For its derivation, see J.G. Charney, J. Meteorology 6 (1949), 371-85.

The objective of the numerical experiments was to see how well the resulting calculations predicted the evolution in time of observed isobars. These isobars and their correlation with cyclones and anticyclones constitute one of the most conspicuous features of weather maps.

Shock and Blast Waves. From 1941 on, von Neumann was concerned with shock, blast, and detonation waves, primarily in military contexts. Especially during the war years of 1942-45, I watched him pore over spark shadowgraph pictures of projectiles in flight, and got a vivid impression of his interest in mathematical formulations and results, and of his realism and skepticism as to their validity. Some idea of the scope and content of his work in this area can be gleaned from his reports and papers, collected in [24, Vol. VI] as Items ##19-29.

His initial studies (##19-26), made in 1941-43, used analytical methods to predict: (a) "the laws of decay of a blast wave due to a point explosion of energy E_0" using "the so-called similarity property of the solution".* (b) "the origin of explosions and the propagation of their effects" in plane shock waves including "when the so-called Chapman-Jouguet hypothesis is true, and what formulas are to be used when it is not" (##19, 20).†

Brilliant, essentially algebraic studies of interactions between two or more plane shock waves were a by-product of the studies of (b); ##22-23 describe this work, which is unrelated to the purposes of the present paper.

*#21, taken from Chap. II of Los Alamos Report LA-200 (1947); see [33, Vol. VI, p, 220] for background dates. The basic ideas stem from G.I. Taylor, Report RC-10 and J. von Neumann Report AM-9, both dated June, 1941. For comparable Russian work, see L.I. Sedov, "Similarity and Dimensional Methods in Mechanics", Academic Press, 1959, Ch. IV, §11.

†Cf. R. Courant and K. Friedrichs, "Supersonic Flow and Shock Waves," Interscience, 1948, §§86-96. For an expert contemporary analysis of the numerical methods then considered, see H. Geiringer, Adv. Appl. Mech. 1 (1948), 202-48.

Particle Models. By 1944, von Neumann had already sketched schemes for using high-speed computers to simulate plane shock wave propagation by means of molecular models employing "10 to 100 particles" (#27). In this report, he suggests that "$14^2 \sim 200$ and $14^3 \sim 3000$ [molecules] may be needed in truly two- or three-dimensional problems."

In his later work on plane shocks (#28) with R.D. Richtmyer, published in 1950, and on spherical shocks (#29) with H.H. Goldstine, published in 1955,* he used now standard difference approximations and "artificial viscosity" to smooth out the calculations. Since this work and subsequent extensions of it have been authoritatively analyzed by Richtmyer and Morton in [28, Secs. 12.8 - 12.13], I shall only say that advances since von Neumann's work have not represented improvements by orders of magnitude; cf. §11 below.

4. Von Neumann's Legacy, II. I shall conclude my brief summary of von Neumann's legacy to "numerical hydrodynamics as a mathematical science" by calling your attention to three other papers.

Petroleum Reservoir Exploitation. The first two of these [33, Vol. V, ##19-20] are concerned with petroleum reservoir exploitation, and were written for the Standard Oil Development Company in 1947-48. They dealt with parabolic systems, and included a far-reaching generalization of the classic Courant-Friedrichs-Lewy analysis of the stability of difference approximations. This was presented at the Naval Ordnance Laboratory in August 1947, and published by G.C. O'Brien, M.A. Hyman and S. Kaplan in J. Math. Phys. 29 (1950), 223-39.†

*The preceding papers were reprinted from J. Applied Phys. 21 (1950), 232-7, and Comm. Pure Appl. Math. 8 (1955), 327-53. For the latter, cf. Sedov, op. cit. supra, and G.I. Taylor, Proc. Roy. Soc. A201 (1950).

†R.S. Varga and I developed it further in [7].

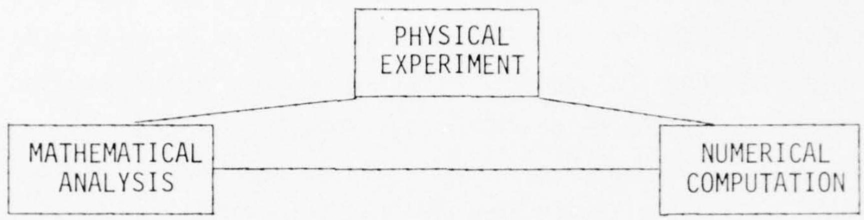
These reports give a vivid idea of the primitive state of computing at the time. Von Neumann began by comparing an estimated cost of 12.5¢ per multiplication for a trained operator using a desk computer, with a cost of 1.4¢ on the IBM SSEC calculator [33, Vol. V, p. 665]. Nine years later, I made similar estimates independently [7, pp. 205-6], my corresponding cost estimates were 2.5¢ and .002¢, based on my experience with more efficient Harvard graduate students and "second-generation" computers. Von Neumann concludes by describing a "small" problem as having 10 Δx intervals and 20 Δt intervals, and a "large" problem as having 25 Δx intervals and 120 Δt intervals [33, Vol. V, p. 750]. At least there has been progress in some areas over the past 27 years!

Turbulence. Finally let me remind you that von Neumann gave much thought to turbulence. He collected his thoughts in a long manuscript [33, Vol. VI, #33], which he never polished for publication. He concludes on a somewhat pessimistic note, stating in part that [33, Vol. VI, p. 469]:

> "there might be some hope to 'break the deadlock' by extensive, but well-planned computational efforts. It must be admitted that the problems in question are too vast to be solved by a direct computational attack, that is, by an outright calculation of a representative family of special cases. ... however, ... one could name certain strategic points ... where relevant information must be obtainable by direct calculations. If this is properly done, and the operation is then repeated ..., etc., here is a reasonable chance of effecting real penetrations into this complex of problems and gradually developing a useful, intuitive relationship to it. This should, in the end, make an attack with *analytical* methods, that is truly more mathematical, possible."

I have here italicized the word "analytical" for emphasis; von Neumann was always mindful of there being three different tools of scientific investigation: physical experiment, mathematical analysis, and numerical computation. He was also very much aware that any one of these might provide the clue to a new scientific insight, but that all

three should give consistent quantitative results before a scientific theory can be accepted as valid. The following diagram is intended to stress the importance of these realizations.



5. Current Computing Costs. Von Neumann's achievements are especially impressive when we consider the limited capabilities of computers in the late 1940's and early 1950's. In von Neumann's time, only 10^3 - 10^4 multiplications could be carried out for a dollar, today, one can do 10^6 - 10^7. Hence for $1000, a good computer will perform 10^9 - 10^{10} arithmetic operations, a number which staggers the imagination. It is the number of seconds in three centuries!

Nevertheless, even 10^{12} operations do not suffice for many computations in fluid mechanics. Thus, suppose we wish to solve an initial value problem for fluid flow in a bounded region of three-dimensional space. To describe the details of the flow adequately at any given instant t, it may barely suffice to know the velocity components u,v,w, the pressure p and the temperature T at each meshpoint of a $10\times30\times40$ mesh. This gives a total of 60,000 numbers, to store which even in single precision at one time-step uses up a large fraction of the core storage of an IBM 370/165 computer.

To predict the evolution of such a compressible flow in time using a "marching process" may typically take about 100 arithmetic operations per unknown per time step, and so 6×10^6 operations per time step for the

mesh described above. To perform these on an IBM 370/165 takes about 6 seconds and costs about \$1; hence only 1000 time steps can be taken for \$1000 under the above conditions. Projected graphically as a movie at 60 cycles per second, the show would be over in 15 seconds! One naturally wonders which is a more powerful experimental tool: a computer or a camera?

With incompressible flows, one cannot in general predict their evolution in time by a simple marching process, because the "state" of the flow at any one meshpoint is apt to be indirectly influenced by that at all others, even though the difference approximation used relates values at a meshpoint only to those at adjacent meshpoints.

In the cases of potential flow and plane viscous flow to be discussed below (§6 through §9), one can fortunately express the condition of incompressibility by linear equations ($\nabla^2\phi = 0$ and $\nabla^2\psi = \zeta$, respectively). However, in the three-dimensional case and with an $n\times n\times n$ mesh and hence $N = n^3$ meshpoints, the coefficient-matrix of the resulting linear system (assuming this favorable case) will have band half-width about n^2. To solve such a system directly by band-elimination requires fillin of about n^5 numbers per dependent variable. To store this many numbers in core in (say) the 370/165 is impossible for $n \leq 10$. The solution of such a small problem by elimination will then take at most $n^4N = n^7$ operations, and so be relatively cheap.

The preceding storage limitation is much too severe to permit the accurate detailed treatment in core of really complicated three-dimensional flows (see §15), by direct methods. Using iterative methods, one can treat somewhat larger meshes in core, but even so and allowing only single precision and one unknown per meshpoint (e.g., the velocity potential ϕ), a $40\times25\times20$ mesh represents about the upper limit. Using previous values to give a good initial guess at each time-step, about 100 iterations should suffice to get reasonable accuracy, giving an operation count of $10^7 - 10^8$ per time step.

The estimates given above, assuming partial difference approximations, are extremely rough, but even allowing an uncertainty factor of 10 or 100, it should be apparent that the power of current computers is strained by complicated three-dimensional problems in fluid dynamics. Estimates based on finite element models would be not too different. Moreover with integral equation methods, even those for single or double distributions on surfaces such as arise in classical potential theory, to store the kernel $k(s,t,s',t')$ for (s,t) and (s',t') on the surface of an $n \times n \times n$ domain, hence for about $6n^2$ boundary points, will require about $36n^4$ numbers and again exhaust the storage of an IBM 370/165 if $n^4 > 1600$ (i.e., if n exceeds 6).

POTENTIAL FLOWS

6. General Remarks. So far, the discussion has related to fluid dynamics in general -- except for its concern with von Neumann's special interests. From here on, I shall emphasize problems having especial naval interest, and therefore speak primarily of hydrodynamics, rather than aerodynamics. It is well known that, in an ideal incompressible, non-viscous fluid without flow separation, initially at rest, the velocity (u,v,w) must be the gradient of a velocity potential ϕ, so that

$$(6.1) \qquad u = \partial\phi/\partial x, \quad v = \partial\phi/\partial y, \quad w = \partial\phi/\partial z.$$

It is generally agreed that many real fluid motions in water is well approximated by the preceding equations outside of boundary layers (where vorticity is generated) and wakes (into which the boundary layer and hence vorticity is fed). Furthermore [1, p.9], this velocity potential must satisfy the Laplace equation

$$(6.1') \qquad \nabla^2\phi = 0,$$

and the Bernoulli equation

$$\frac{\rho}{2}\nabla\phi\cdot\nabla\phi + \rho\, \partial\phi/\partial t + p = P(t) + \rho g y, \tag{6.2}$$

where P(t) is some ambient stagnation pressure level. Furthermore, the potential flows defined by Eqs. (6.1) and (6.2) are the fluid flows most amenable to analytical treatment, as in Lamb [24], Chs. III - VI and VIII - IX.

Moreover such potential flows, bounded by fixed surfaces and free surfaces at constant pressure, include many flows of great interest to the Navy. Thus they include the motion of gravity waves, much of the flow around a ship hull (and the resulting wave resistance of ships), and many cavity flows (see [8, Chs. I-X] and my review article in [31, pp. 19-38]). For this reason and because of the simplicity of the underlying mathematical model defined by (6.1)-(6.2), it would seem desirable to include simulations of such potential flows in any systematic attempt by our Navy to develop a capability for numerical hydrodynamics.

At the present time, codes based on the MAC method seem most nearly ready to solve such problems. Some illustrations of its capabilities are described in [32]; other illustrations are reviewed briefly in [31, p. 33]. However, MAC codes do not incorporate incompressibility or irrotationality as constraints,* but achieve them approximately by iterating. Also, an "artificial viscosity" is introduced in MAC codes to smooth out the free surface, and a zero gradient in tentative velocities at the boundary [31, p. 238, top] is imposed somewhat arbitrarily just because "it is found to have little upstream effect."

S.A. Orszag [27, pp. 289-93] has used quite different "spectral methods", in which exact potential flows are combined by superposition, with coefficients computed so as to make prescribed boundary conditions nearly satisfied. Though such spectral methods seem promising for gravity wave studies, they have not been carefully tested on three-dimensional problems, or on problems involving moving solid boundaries.

*I.e., as exact conservation laws for volume and circulation, which are exact invariants in the model (6.1).

For these reasons, it would seem desirable to try out more conventional difference approximations to (6.1)-(6.2), to compute solutions of potential flow problems. The application of such methods will have one disadvantage: solutions $\phi(x,y,z;t)$ must be differentiated to obtain the velocities as in (6.1) and numerical differentiation introduces a substantial loss of accuracy. However, one can hope to reduce this loss by using methods having an $O(h^4)$ order of accuracy in ϕ, like the one proposed in Appendix A below.

In this connection, it seems relevant that Kreiss and Orszag have independently advocated the use of higher-order methods in fluid dynamic calculations.* (See also Roache [29, Sec. III-A-19].)

7. Gravity Waves. Gravity waves, whether caused by ships, submarines, or wind, are of especial naval interest. So are tides, and it is not surprising that Lamb should have devoted two long chapters to their study [24, Chs. VIII and IX].

Moreover one of the easiest kinds of potential flows to simulate, analytically or numerically, is the family of two-dimensional gravity waves in shallow water. Here the Neumann condition $\partial\phi/\partial n = 0$ simulates the effect of rigid walls, and the free surface condition $p = \text{const.}$ that of the air. It would seem very desirable to apply the MAC and related methods to such flows, to study their cost effectiveness, and to subject their accuracy to a searching theoretical and experimental (numerical experiments) analysis.

Care must be taken to specify the physical problem intended -- e.g., as the evolution in time of an initial local disturbance in a fluid otherwise at rest, or as the passage of a train of periodic waves; otherwise

*See H.-O. Kreiss and J. Olger, Tellus 24 (1972), 199-215; S.A. Orszag, J. Fluid Mech. 49 (1971), 75-112, and J. Comp. Phys. 14 (1974), 93-103.

the flow cannot be determined. Moreover even for well-determined problems, it is hard to avoid spurious "end effects" or to simulate "deep water" waves without using a very large domain -- and hence being very wasteful computationally.

Some calculations of gravity waves in shallow water have been reported in Nichols and Hirt [26, p. 242]. It would be interesting to compare these calculations, which seem to have $O(h^2)$ accuracy in u and v, with calculations based on the approximation described in Appendix A, which aims at achieving $O(h^4)$ accuracy in ϕ and hence $O(h^3)$ accuracy in velocity.

In the MAC method, as was mentioned in §6 and is explained in Roache [29, pp. 194-5], it seems to be essential for computational stability to retain a "dilation" term $D = u_x + v_y$, even though this is nominally zero. A superficial study of the literature suggests that, for two-dimensional gravity waves, it may be better to use the stream function $\psi(x,y;t)$ as unknown. This also satisfies the Laplace equation $\nabla^2\psi = 0$, and $\nabla\psi\cdot\nabla\psi = \nabla\phi\cdot\nabla\phi$. However, the <u>free surface</u> condition seems harder to formulate in terms of ψ.

<u>Poisson Equation Solvers</u>. To solve the Laplace equation for $\partial\phi/\partial t$ from the boundary condition (6.2) is more costly than solving the Poisson equation on a rectangle for the boundary conditions $\psi = 0$. With a 63×63 mesh, the latter takes about 0.5 seconds and costs about 10¢ on the IBM 370/165 at MIT. Whereas to solve the Dirichlet problem on the same mesh takes about 5 seconds using the SL-MATH band-matrix subroutine; hence the cost might be $1 per iteration. (For the 68×10 mesh used by Nichols and Hirt, the cost would presumably be much less.)

<u>Finite Element Models</u>. The use of finite element models in solid and structural mechanics is a highly developed art, which has largely displaced the use of difference approximations. It is natural to wonder how useful they might be in fluid mechanics. I shall make various technical remarks on this question below, developing some of them in appendices.

Firstly, it should be realized that finite element methods are most naturally formulated for steady state problems (i.e., differential equations of elliptic type associated with a variational principle). In time-dependent problems, they lead most naturally to semi-discretizations, which are equivalent to large systems of stiff ordinary differential equations. Although general purpose codes for integrating such systems have recently been developed by Gear, Hindmarsh, Shampine, Hull and Enright, Madsen and others, their suitability for integrating particular stiff systems arising in the way just mentioned has not been carefully tested.

A second remark, stemming from Courant, is that many finite element approximations are naturally associated with difference approximations. Thus V. Dougalis and the author have shown that,* if one uses the 5-point approximation to the Laplace equation on the staggered square mesh most commonly used in fluid flow problems, then compatible conjugate discrete harmonic functions ϕ and ψ are defined. One can then interpolate by piecewise bilinear, hence piecewise harmonic finite elements. However, the application of variational methods to these interpolants gives 9-point formulas, and discontinuous $\nabla\phi$ and $\nabla\psi$, hence discontinuous u and v. I have not yet seen any good way to either (a) construct useful higher-order harmonic elements, or (b) to use the model just described to solve particular problems.

Time-Dependent Free Surface. One of the most ticklish problems consists in (a) constructing step-by-step the free surface corresponding to difference or finite element approximations, and (b) suitably approximating the Laplace equation on the "irregular stars" formed by the boundary and adjacent meshpoints. A good test problem would be the case of the

*G. Birkhoff and V. Dougalis, "Approximations to the Laplace Equation," unpublished MS.

stationary potential flow (with surface wave) caused by a solid cylinder moving at constant velocity below the free surface, normal to its axis -- either in shallow or deep water. Numerical solutions could be compared with Havelock's exact analytical solution.*

8. Ship Waves. The advantage of numerical methods over analytical methods in treating potential flows becomes much more pronounced in the case of two-dimensional gravity waves, of the kinds treated analytically by Lamb in his Arts. 255-9. Among these, ship waves are of especially great naval interest as the cause of ship wave resistance.

Because of this interest, many ingenious analytical methods were developed by Michell, Havelock,† and others for calculating the wave resistance by purely mathematical methods.

As von Neumann often observed, analytical methods are usually best suited to linear problems, and hence (in the present context) to waves of small amplitude caused by "thin" ships. Numerical methods depend far less on linearity. Hence with modern computers, it should be possible to calculate reasonably economically the (theoretical) wave resistance of "full" ships causing waves of finite amplitude (e.g., the bow wave). Of course, in this case as for "thin" ships, the total resistance is not simply the sum of the calculated wave resistance and the measured wake ("eddy") and skin friction resistances.‡ However, the "scaling up" of model tests of the total drag measured in towing tanks is also more of an art than a science.

---

*T.H. Havelock, Proc. Roy. Soc. A157 (1936), 526-34.

†See "The Collected Papers of Sir Thomas Havelock on Hydrodynamics," edited by C. Wigley, Publication ONR/ACR-103, U.S. Office of Naval Research, 1964.

‡See [8]; also R.G. Strandy, J. Fluid Mech. 62 (1974), 625-42.

With numerical methods, it should not be too much more difficult to calculate the effect on ships of waves (periodic or random) in a model of a seeway, in the potential flow approximation. Of course, the physical significance of such calculations would have to be inferred from other model experiments and full-scale observations.

Seiches. Another interesting class of potential flows arises in connection with mathematical models of "seiches", or long-period waves in lakes, harbors, etc., such as are treated in Lamb, Arts. 185-93, 209-11 and 257-9. It should be relatively easy to calculate these on a computer, and to check with the analytical theory in case of constant depth, where it suffices to find the eigenfunctions of two-dimensional Helmholtz equation. One great advantage of numerical methods is their easy adaptability to the case of variable depth, and to estimating the amplification of wave steepness by shallow water.

Unfortunately, there seems to be no reason to believe that the dissipation of storm waves in harbor basins after repeated reflections from breakwaters can be realistically modeled by potential flows. General energy considerations would seem to be equally reliable for predicting this.

9. Interfacial Discontinuities. Many fluid flows can be represented approximately as two potential flows in adjacent regions, separated by a free boundary at constant pressure, or other flexible interface across which the velocity changes discontinuously. Such potential flows provide the standard models for cavitating flows, the generation of waves by wind, and other manifestations of Helmholtz and Taylor instability.

I reviewed the mathematical theory of cavitation in 1971 in [31, pp. 19-37] and do not have much to add to what I said there. However, I would like to call attention to the calculations reported by Paul Garabedian in the same volume [31, pp. 197-9], of the axially symmetric Riabouchinsky cavity flow past a circular disc. He reported that the

computing time was about one minute per case on the CDC-6600. It would doubtless have been very much longer, and convergence much harder to achieve, had he not neglected gravity or even treated an obstacle that was not axially symmetric.

This remark is just one example of the most important shortcoming of potential flow calculations: their lack of physical realism under many circumstances. Thus, even after many years of ingenious research by very eminent scientists, the physical mechanisms responsible for the generation of ocean waves by wind are far from clear. We simply do not have a well-defined mathematical model, for either analytical or numerical treatment. I shall return to this question in §15.

OTHER HYDRODYNAMICAL MODELS

10. Acoustic Waves. After potential flows, the most tractable problems of fluid motion should be those concerning sound waves of infinitesimal amplitude (acoustic waves) in a homogeneous fluid. They should be tractable because the associated pressure perturbation $\delta p = u$ satisfies the relatively simple wave equation

$$u_{tt} = c^2\nabla^2 u = c^2(u_{xx} + u_{yy} + u_{zz}). \tag{10.1}$$

The "progressive wave" solutions of this equation (and its variable coefficient analogues) are of great naval interest in connection with sonar; the "standing wave" solutions of the form

$$\delta p = P(x,y,z)e^{i\omega t}, \tag{10.2}$$

and satisfying the (Neumann) boundary condition $\partial P/\partial n = 0$, are also of long-standing scientific interest. It is classic (and obvious) that δ_p satisfies (10.1) if and only if P satisfies the Helmholtz or "reduced wave" equation

$$\nabla^2 P + k^2 P = 0, \quad k^2 = \omega^2/c^2. \tag{10.3}$$

Many other acoustic problems of engineering interest are treated analytically in [25]. These include the behavior of condenser microphones (Secs. 20, 30), the propagation of sound in tubes and horns (Secs. 23, 24), the scattering of plane waves from an obstacle of general shape (Sec. 29), the determination of the normal modes of vibration of a "room" of general shape (Sec. 32), and the response of an air space to forced vibrations (Sec. 34). In many cases, one can use exact analytical solutions for special geometries (such as the cylinder and sphere) to test the accuracy of benchmark numerical calculations.

The author and George Fix have published a long paper [36, Vol. II, pp. 111-51] on numerical methods for computing the eigenvalues of (10.3) in simple geometries.

V. Dougalis and the author have written a report [7'] analyzing various difference and finite element schemes for solving (10.1) in one and two space dimensions. Although there are surprisingly few published numerical experiments treating two-dimensional problems, we are optimistic about the possibility of computing the propagation of sound waves of general shape for many wave lengths in free space and in straight channels. However, Professor Calvin Wilcox of the University of Utah has warned us that he has found it much more difficult to reproduce on a finite mesh the initiation of scattered waves from <u>curved boundaries</u>.

Having developed satisfactory techniques for treating numerically any of the problems listed above in a given geometrical configuration, it should not be too hard to extend them to include the effects of <u>dissipation</u> [8, Sec. 33]; are also [24, Arts. 358-364], since this tends to "smooth out" solutions.

Another challenging problem concerns sound propagation in a stratified medium. However, if one is only interested in the way in which

sound energy is transmitted, it may be hard to find numerical methods as accurate as those corresponding to ray-tracing.

11. Explosion Waves. Actually, Eq. (10.1) is just a linearization of a much more complicated system of nonlinear equations for the adiabatic motion of a gas. If gravity is neglected, these are still compatible with the existence of a velocity potential (i.e., irrotational motion); see [24, Ch. X], which is devoted entirely to irrotational "waves of expansion," which include sound waves (see §11) and explosive waves. I shall discuss here only the simplest case of plane pressure waves of finite amplitude, studied by Riemann. In this case, the equations of motion simplify in so-called Lagrangian coordinates as in [1, (20)] and [24, Arts. 281-4] to

$$x_{tt} = - H'(x_a)\, x_{aa}. \tag{11.1}$$

As the simplest mathematical model for explosion waves, Eq. (11.1) and its cylindrical and spherical analogs fascinated von Neumann; I have tried to summarize his contributions to its numerical solution in §3. As Stokes already knew, exact solutions of (11.1) ordinarily develop shock discontinuities in finite time, (rarefaction waves are an exception).†

It was evident to von Neumann and all later workers that such discontinuities and even steep fronts would be hard to locate precisely by difference methods, in either Eulerian or Lagrangian coordinates, and that this would make it difficult to solve Eq. (11.1) accurately by any numerical method on a discrete mesh. Indeed, artificial irregularities propagate from shock discontinuities if the most natural difference approximations are used (see [28, §12.13]); it was to damp these out

---

†For a thorough analysis of this phenomenon, see the treatise "Supersonic Flow and Shock Waves" by R. Courant and K. Friedrichs, Interscience, 1948.

that von Neumann and Richtmyer introduced an "artificial viscosity" (see §3, end). *This did not simulate the real thickness of the shock, which* is a fraction of the mesh length, but kept it down to a few mesh lengths.

Many attempts have been made to perfect the technique just described; two relatively good procedures, one of them due to Lax and Wendroff, are reviewed in [28, Secs. 12.14 and 12.15].* However, it cannot be said that any published method gives a truly accurate solution of Riemann's initial value problem. Patrick Roache has summarized the situation as follows:†

> "It is shown that the usual analysis for the implicit artificial viscosity of finite difference analogs of the linear advection equation is ambiguous, with different results being obtained for the transient and steady-state solutions. It is demonstrated that the currently most popular methods, touted as having no artificial viscosity, actually do have such when applied to steady-state problems."

A similar negative judgment was given by Moretti in 1971; he wrote:‡

> "Is a computer program available [for solving one-dimensional compressible flow problems, which is], easy to use, general, safe, accurate and fast? *Judging by the existing literature and requests* from...industry, the answer seems to be...negative."

Since that time, Moretti has himself developed a new "shock-locating" technique‡‡ which seems to be more powerful than any of its predecessors; we are currently testing it at Harvard.

---

*See Peter Lax, SIAM Rev. 11 (1969), pp. 7-19, for a general overview of the situation as he sees it.

†J. Comp. Phys. 10 (1972), 169-84; see the Abstract and Summary. This paper is reproduced on pp. 351-66 of Roache's book [29]. Another thoughtful recent paper is by B.S. Masson, J. Comp. Phys. 10 (1972), 88-102.

‡G. Moretti, "Complicated one-dimensional flows," Brooklyn Polytech.Inst., Sept 1971. See also G. Moretti and M.D. Salas, J.Comp. Phys, 5 (1970), 487-506.

‡‡See next page.

Burgers-Hopf Equation. I have not tried to review here the status of attempts to solve numerically the Burgers-Hopf equation, variants of the Korteweg de Vries equation,* or the Fisher-Kolmogoroff equation. Suffice it to say that there is a large literature concerning at least the first two of these, and that shock near-discontinuities plague these models as viscosity tends to zero in much the same way that they plague attempts to solve the Lagrange piston or other shock problems.

Supersonic Flows. Much of the research on shock waves has been done in connection with supersonic flows; see the footnote on page 24. In particular, many calculations have been made of compressible flows around airfoils [29, Chs. IV-V]. Though such calculations refer to air, and are therefore only indirectly relevant here, I do want to call attention to [37]. This is a landmark from the scientific standpoint, because it treats everything from mathematical theory to computer programs.

12. Navier-Stokes Equations. The mathematical models of fluid flow referred to above all neglect viscosity. Although this is exceedingly small in water, it is well known to be the direct cause of skin friction and the indirect cause of eddy resistance. This and the next two sections will be concerned with numerical models of fluid flow which include real viscosity as a physical entity. The main thrust of my comments will be that much more applications-oriented basic research is needed to make such models reliable.

---

‡‡G. Moretti, "Thoughts and afterthoughts about shock computations," Brooklyn Polytech. Inst., Dec 1972. His technique is sometimes called "shock-fitting", as contrasted with the "shock-capturing" (or shock smearing) technique based on artificial viscosity.

*See Brooke Benjamin, Lectures in Applied Mathematics, Vol. 15 (1974), 3-47. For a numerical scheme, see D.H. Peregrine, J. Fluid Mech. 25 (1966), 3-47.

I shall restrict attention to <u>incompressible</u> viscous flows for three reasons. First and most important, the dynamic effect* of compressibility is negligible at Mach numbers $M = |u|/c << 1$, where c is around 5000 ft/sec in water. Second, and more philosophically, there are some uncertainties connected with bulk viscosity: it is not clear precisely what equations for compressible viscous flow should be assumed [1, p. 47]. And third, the computational difficulties are quite big enough without making the differential equations to be solved more complicated.

Actually, the assumption of incompressibility introduces its own computational problems. It implies instantaneous <u>action at a distance</u> (instead of pressure waves transmitted at 5000 ft/sec). Since the motion of every point influences every other point immediately, the computation of time-dependent incompressible flows requires solving a large system of simultaneous linear equations at each time step!

In two dimensions (i.e., for incompressible <u>plane</u> viscous flows), this difficulty can be minimized by using the (scalar) <u>vorticity</u> $\zeta = \partial v/\partial x - \partial u/\partial y$ as a dependent variable. One then has the equations

$$(12.1) \qquad D\zeta/Dt \equiv u\partial\zeta/\partial x + v\partial\zeta/\partial y + \partial\zeta/\partial t = \underline{u} \cdot \underline{\zeta} + \nu\nabla^2\zeta.$$

Knowing the vorticity, the stream function ψ can be determined, the then new values of u and v computed, from the equations

$$(12.2) \qquad -\nabla^2\psi = \zeta, \quad U = \frac{\partial\psi}{\partial y}, \quad v = -\frac{\partial\psi}{\partial x}.$$

Hence at each time-step, in principle, one can use first (12.1) to predict a new vorticity distribution, and then (12.2) to predict the new velocity-distribution.

*For sound waves, on the other hand, compressibility is the essential factor.

A large number of schemes for solving two-dimensional time-dependent problems governed by Eqs. (12.1)-(12.2) have been tried, especially by Leith, Arakawa, O'Brien and others in connection with weather-forecasting models. I shall not try to supplement Roache's review of this work in [1, pp. 15-204], except to say that it seems still to be very expensive to solve any but the simplest problems.

Time-independent Flows. Recently, H.B. Keller and others* have made systematic computations of 'steady' (i.e., time-independent) incompressible, viscous flows past cylinders and spheres for Reynolds numbers ranging from 0.1 to 200. In the notoriously difficult case of a cylinder†, results consistent to about ±5% were obtained. In the case of solid spheres, agreement 'to three or four digits' with earlier work of Dennis and Walker (J. Fluid Mech. 48 (1971), 771-789) is reported; for flows around spherical gas bubbles, discrepancies of 5% and more are reported.

13. Laminar Boundary Layers. As the Reynolds number $Re = u_{max}\ell/\nu$ increases, it becomes increasingly difficult to integrate the Navier-Stokes equations numerically. This is for two reasons. First, the vorticity becomes concentrated in a very thin boundary layer whose thickness δ is very small in comparison with the length scales of interest. Thus, we are again faced with the curse of discontinuity, already discussed in connection with shock waves in §11, and with cavitation in §9. And second, the flow becomes very irregular, or "turbulent".

To illustrate the first difficulty, consider a ship 200 meters long moving at a speed u = 10 meters/sec (less than 20 knots). The

*F. Nieuwstadt and H.B. Keller, Computers and Fluids 1 (1971), 59-71; D.C. Brabston and H.B. Keller, J. Fluid Mech. (to appear in 1975).

†See [1, Secs. 30-31]; also the discussion of Fromm's calculations in the next section.

displacement thickness δ of the laminar boundary layer predicted by the Navier-Stokes equations is given approximately [18, p. 136] by the formula

$$\delta = 1.72(\nu x/U) \approx .05\sqrt{x}. \tag{13.1}$$

since $\nu \approx 10^{-2}$ in water (c.g.s. units) and $U = 1000$ cm/sec. Hence for $x = \ell = 200$ meters, the displacement thickness of a laminar boundary layer would be about 7 cm. To calculate accurately the thickness of this boundary layer by direct numerical methods would require locating at least 5 mesh lines within the boundary layer, spaced therefore only 1-2 cm apart - or $10^{-4}\ell$, where ℓ is the length of the ship.

Since the wave field needed to estimate either the ship wave resistence or the effect of storm waves on the ship's motion must extend several ship lengths to make the calculations of §8 adequate, we see that to include even laminar viscosity effects, we would have to use length scales differing by several orders of magnitude. Furthermore, to provide a realistic substitute for towing tests, the calculations would have to apply to turbulent boundary layers (see §15) along two-dimensional surfaces of a general shape, and correctly predict the onset of separation (see §14).

State of the Art. Although substantial recent advances seem to have been made by H.B. Keller and T. Cebeci,* boundary layer calculations up to now seem to have been successful at reasonable expense only for one-dimensional boundaries, and then for 'steady' flows. Finally treatments of turbulent boundary layers are highly empirical, and are not based on the Navier-Stokes equations. Hence they are in no sense part of 'numerical hydrodynamics as a mathematical science.'

*See J. Comp. Phys. 10 (1972), 151-61; Springer Lecture Notes in Physics No. 19 (1973), 79-85. A review paper by Keller will appear in Proc. Fourth Int. Symp. Numer. Fluid Dynamics, Springer, 1975.

Separation. The need for calculating the velocity profile within the boundary layer reasonably accurately becomes especially clear when we realize that flow separation is very sensitive to the velocity profile within the boundary layer, occurring according to Prandtl's theory [18, pp. 57-58] precisely when its normal derivative $\partial u/\partial n$ vanishes on the surface. And the form drag of a body and wake behind it are determined largely, to a first approximation, by the point where separation occurs. Indeed, this is precisely why so much research has been devoted to boundary layer control (see [18, Ch. XII] and [30, Ch. XIII]).

Unfortunately, exact mathematical analysis has so far proved incapable of predicting form drag. Aerodynamic design has been largely empirical, and the role of mathematics (analytical or numerical) has been limited to coordinating simple empirical engineering approximations. Books like [18] and [30] do not really treat hydrodynamics (or aerodynamics) as a mathematical science (in the sense of Lagrange) at all!

Wakes. It would be most desirable to be able to predict not only separation but flows in the wakes behind solid obstacles, thus revitalizing the vision of Lagrange. The most promising approach would seem to be to use computers, in the spirit of von Neumann, to integrate suitable approximations to the boundary layer and Navier-Stokes equations. A first objective would be to predict accurately the observed flow separation and wake behind a number of relatively simple cylindrical bodies (circular cylinder, square cylinder, etc.), held normal to the flow.

Following an old idea of Rosenhead, several people, including the author ([2] see also [35, Vol. XIII, pp. 73-74]), have tried to approximate the evolution of two-dimensional vortex sheets shed by solid cylinders by neglecting ordinary diffusion and (following Kelvin!) considering only

*Prandtl's boundary layer equations represent, of course, just a simplified asymptotic approximation to the Navier-Stokes equations; see [18, Secs. 44-45] and [30, Ch. VII].

the mutual interaction of point-vortices. Recently some very interesting numerical experiments based on this idea have been performed by A. Chorin.*

The most impressive calculations of this kind to date are those carried out by J.E. Fromm ([16], [17]) with F.H. Harlow at Los Alamos. They simulated vortex streets behind square cylinders very plausibly. However, these calculations were never critically compared with observation, to verify the accuracy with which they reproduced the many sensitive phenomena associated with vortex streets (wall effects, surface roughness, and critical Reynolds numbers to mention a few); see [8, Ch. XIII, §§8-9]. Actually, Louis Rosenhead has suggested that vortex streets may be unstable to three-dimensional disturbances, and hence their evolution may not be predictable by <u>any</u> two-dimensional model!†

In view of all these facts, a deeper study and critical comparisons with physical experiments of outputs from the impressive programs developed by Fromm would seem to provide the best immediate prospect for testing the reliability of numerical calculations of laminar boundary layers and associated flow separation.

14. <u>Turbulence</u>. It is well known that the boundary layer of a stream flowing past a flat plate ordinarily becomes <u>turbulent</u> when the local Reynolds number $Re = U\delta/\nu$ exceeds 2000-5000. By the formula of the preceding section, this implies that the boundary layer will ordinarily become turbulent when $U\ell/\nu$ exceeds $10^5 - 10^6$ [18, Sec. 151]. For even a smooth ship (e.g., no barnacles) cruising at 10 meters/sec, therefore, the boundary layer may be expected to become turbulent a foot from the bow. Since the onset of flow <u>separation</u> is greatly delayed by turbulence (this is what the "critical Reynolds number" is all about!), computational

*A.J. Chorin, Math. Comp. 22 (1968), p. 745 and J. Fluid Mech. 57 (1973), p. 785.

†L. Rosenhead, Proc. Roy. Soc A129 (1930), 115-35; Adv. Appl. Mech. III (1953), p. 191.

models which ignore this turbulence are sure to be unreliable. Something must be done that is analogous to the roughening of towed small-scale ship models to simulate turbulence.

The difficulty of predicting, or even describing mathematically the velocity fields in flows containing turbulent boundary layers is evident: there may be scores or even hundreds of small vortices to be modelled in each few cc. of water. For a ship hundreds of feet long, this would require specifying the velocity at literally millions of meshpoints. This far exceeds the storage capacity of current computers; moreover to integrate the resulting discretized equations of motion would be prohibitively expensive.

Atmospheric Turbulence. An even more impossible problem in exact mathematical description arises in atmospheric turbulence. The extreme complexity of physical reality is perhaps best illustrated by clouds, with their all-important convection currents, latent heat (of vaporization), and possible electrical discharges (lightning), not to mention the influence on condensation of smoke and dust particles (or AgCl crystals) used in "cloud-seeding". It is evident that to characterize the turbulent velocity field in even one cubic mile in sufficient detail to predict its future evolution would require millions of meshpoints. Since there are 10^{12} (a quadrillion) cubic miles in the earth's atmosphere up to a height of five miles, we see that an adequate direct attack on the numerical integration of the Navier-Stokes equations is very far out of range. To achieve worldwide weather prediction by this direct approach verges on the hopeless.

Recently, there have been ambitious efforts to treat oceanic circulation by methods analogous to those used in attempts at numerical weather prediction.* It seems relevant to point out that efforts to do this

*See George Fix, "Finite element models for ocean circulation problsms," to appear in SIAM J. Applied Math., 1975; and "Finite elements and fluid dynamies," MS.

rigorously are obstructed by similar, if less extreme complications. Especially, turbulence (random vorticity) is ever present; wind friction may be an important driving force; and variations in ocean depth are relatively greater than those in atmospheric thickness. Though dust and dissolved gas (the analog of humidity) may be negligible, salinity and temperature effects (the latter influenced by the cloud cover) may be significant.

The preceding mathematical difficulties have, of course, been well known by experts for a very long time;* von Neumann was well aware of them, and wrote that "good resolution cannot be expected for any but the largest scale motions" [33, Vol. VI, p. 420].

Primarily for this reason all existing numerical simulations of meteorological conditions rely heavily on statistical models of turbulent flow. Several such models are supported by plausible intuitive arguments and empirical evidence.† However, they certainly do not treat numerical weather prediction as a mathematical science.

Statistical Averages. There is an obvious analogy between turbulence and the molecular constitution of gases. This analogy was already discussed at length by Osborne Reynolds, and invoked by Prandtl in his "mixing length" theories of turbulence. It may point the way to a satisfactory mathematical model of turbulence, though all past models verge on pseudoscience.

For gases, the outstanding fact (of which von Neumann was very conscious) is that the macroscopic physics of the order of 10^{24} molecules can be described by ignoring the details of molecular motion, and using a

*G.I. Taylor emphasized them in his classic first paper on turbulence (Phil. Trans. 215 (1915), 1-26).

†See P.G. Saffman, Studies in Applied Math. 53 (1974), 17-34.

handful of simple mathematical formulas expressing the mean pressure p, the mean density ρ, the mean temperture T, and the vector velocity $\vec{u}(x,y,z;t)$ as continuously differentiable functions of position and time. Moreover some of these variables can be eliminated: thus we have $p = f(\rho,T)$, where under normal conditions $f(\rho,T) = c\rho T$ is a good approximation. Hence the state of a gas can be described in terms of two parameters, ρ and T (density and thermal energy). And in adiabatic flow, which approximates gas dynamics well except in shock waves, one can assume that $p = k\rho^{\gamma}$ for a suitable "adiabatic exponent" γ.

This suggests that the "state" of a turbulent fluid can also be expressed in terms of a handful of variables; such as the turbulent "intensity" (or energy per unit volume) and "scale" (analogous to the mean free path). Indeed, these two variables seem to suffice for homogeneous isotropic turbulence, although their evolution in time remains a mystery.

But there are many other kinds of turbulence: boundary layer turbulence, wake turbulence, jet turbulence, convective turbulence, and so on. Although one may hope to describe local conditions in these by deterministic mathematical models involving only a handful of variables (e.g., the components of Reynolds' eddy viscosity tensor $\overline{\rho u_i' u_j'}$) as functions $g_{ij}(x,y,z;t)$, there is as yet no scientific evidence for any such models. My feeling is that the difficulties involved in treating turbulence mathematically are much more like those associated with the kinetic theory of liquids than those of the kinetic theory of gases, and I am not optimistic about substantial progress in the next decade.

15. Two-phase Flows. Even more intractable mathematically than turbulent flows are two-phase flows, such as arise in cavitation, boiling, and condensation. I made a few brief remarks in §9 about the simplest interfacial discontinuities that can arise; I shall now discuss two-phase flows in greater depth.

Discussions of two-phase flows generally accept the following seven variables as indispensible for describing the instantaneous "state" of a two-phase mixture at any point:*

ρ_v, ρ_d the (mean) gas and liquid densities,

u_v, u_d the (mean) gas and liquid velocities,

p the pressure (same in both phases).

T_v, T_d the (mean) gas and liquid temperatures.

They also accept the laws of conservation of mass, momentum, and energy as constraints enabling one to reduce the number of variables by implicit elimination procedures. However, it should be emphasized that the preceding quantitaties are only statistical averages, such as are familiar in turbulence, and that random fluctuations about these averages have a major effect on what actually happens in any specific case. (In mathematical language, the sample space Ω of possible events ω permits wide variations in behavior, depending on ω.)

Thus a casual inspection of photographs of two-phase flows shows that wide variations in mean droplet and bubble size and shape can occur for fixed values of the seven variables listed above, in addition to the local fluctuations mentioned in the preceding paragraph. These have a major influence on the mass and heat "transfer coefficients" ("eddy viscosity and conductivity"), which are typically orders of magnitude greater than the molecular viscosity and conductivity of either phase individually.†

One must also take into account the latent neat of vaporization, the surface tension, and the dissolved gas nuclei and other minute impurities

*See John Meyer, Nuclear Sci. Eng. 10 (1969), 37-63; F. Harlow and A.A. Amsden, "Numerical calculation of multiphase flow," Los Alamos, 1974.

†See William H. McAdams, "Heat Transmission", 3d ed., McGraw-Hill, 1954, esp. Chs. 13, 14.

in order to predict realistically the rates and qualitative nature of boiling (e.g., the possible superheating and supercooling). In these respects, water and liquid metals have quite different characteristics, and it is by no means clear that the approximations which are most appropriate for "modeling" the two-phase flow of water will also be most appropriate for modeling the two-phase flow of liquid metals.

To be specific, there are at least 10-20 "state" variables to be specified at each point, and to predict the probable rate of change of each of these is a function of the current state and its gradient is extremely difficult. To predict the accelerations du_v/dt and du_d/dt is hard enough (they certainly depend on the mean bubble and droplet size and shape); to predict by boiling and condensation is much harder. This is because they depend on empirical coefficients, reliably known only as averages for a few quite special configurations. At the present time, it would seem premature to construct transfer coefficient functions defined over the whole range of the "state" variables, by extrapolation from this limited data.

Before doing this, it would seem wise to compare predictions based on computer models with the empirical data recorded in McAdams, articles in the Journal of Heat Transfer. It would also seem desirable (for reactor applications) to compare such predictions with those given by the four well-tested models used by Meyer (op. cit.) to predict coolant flow in channels, namely:

a. sectionalized compressible model,
b. momentum integral (MIM) model,
c. single mass velocity model,
d. channel integral model.

At the same time, mathematical studies might profitably be made of the problems involved in integrating numerically the resulting partial differential equations.*

REFERENCES

1. Birkhoff, G., "Hydrodynamics: A Stidy in Logic, Fact and Similitude," Princeton University Press, 2d ed., 1960.

2. Birkhoff, G. and J. Fisher, "Do Vortex Sheets Roll Up," Rend. Circolo Mat. Palermo 103 (1959).

3. Birkhoff, G. and G. Fix, "Higher-Order Finite Element Methods," Harvard University, 1975.

4. Birkhoff, G. and S. Gulati, "Optimal Few-Point Discretizations of Linear Source Problems," SIAM J. Numer. Anal.

5. Birkhoff, G., H.H. Goldstine and E.H. Zarantonello, "Calculation of Plane Cavity Flows," Rend. Sem. Mat. Torino 13 (1954), 205-24.

6. Birkhoff, G., B.V. Korvin-Kroukovsky and J. Kotik, "Wave Resistance of Ships," Trans. Soc. Nav. Arch. Marine Eng. (1954), 359-96.

7. Birkhoff, G. and R.S. Varga, "Discretization Errors for Cauchy Problems," J. Math. and Phys. 44 (1965), 1-23.

7'. Birkhoff, G. and V. Dougalis, "Dispersion and Anisotrophy of Discretized Sould Waves," (unpublished MS).

8. Birkhoff, G. and E.H. Zarantonello, "Jets, Wakes and Cavities," Academic Press, 1957.

9. Bramble, J., ed., "Numerical Solution of Partial Differential Equations," Academic Press, 1966.

---

*Some relevant remarks about these problems are made in [9, Part B], and in Ref. 3 of the bibliography of [9, p. 314].

10. Brillouin, L., "Wave Propagation in Periodic Structures," McGraw-Hill, 1946.

11. Courant, R., K.O. Friedrichs and H. Lewy, "Über die partiellen Differenzengleichungen der mathematischen Physik," Math. Ann. 100 (1928), 32-74.

12. Courant, R. and D. Hilbert, "Methoden der Mathematische Physik," Springer, 1931.

13. Daly, B.J., "Numerical Study of the Effects of Surface Tension on Interface Instability," Physics of Fluids 12 (1969), 1340-

14. Davies, R.M., ed., "Cavitation in Real Liquids," Elsevier Press, 1964.

15. Frenkiel, F.E. and K. Stewartson, eds., "High-Speed Computing in Fluid Dynamics," Physics of Fluids Supplement II, Am. Inst. Phys., New York, 1969.

16. Fromm, J.E. and F.H. Harlow, "Numerical Solution of the Problem of Vortex Street Development," Physics of Fluids 6 (1963), 975-82.

17. Fromm, J.E., "The Time-Dependent Flow of an Incompressible Viscous Fluid," Methods Comp. Phys. 3 (1964), 345-82.

18. Goldstein, Sydney, ed., "Modern Developments in Fluid Dynamics," 2 Vols., Oxford University Press, 1938.

19. Harlow, F.H., J.E. Welch, J.P. Shannon and B.J. Daly, "The MAC Method," Report LA-3425, Los Alamos, 1965.

20. Harlow, F.H. and J.E. Welch, "Numerical Study of Large Amplitude Free Surface Motions," Physics of Fluids 9 (1965), 842-

21. Hirt, C.W. and A.A. Amsden, "An Arbitrary Lagrangian-Eulerian Computing Method for All Flow Speeds," J. Comp. Phys. 14 (1974), 227-53.

22. Holt, Maurice, ed., "Proceedings Second International Conference on Numerical Methods in Fluid Dynamics," Lecture Notes in Physics #8, Springer, 1971.

23. Kantorovich, L.V. and V.I. Krylov, "Approximate Methods of Higher Analysis," New York, Interscience, 1958.

24. Lamb, H., "Hydrodynamics," 6th ed., Cambridge University Press, 1932.

25. Morse, Philip M., "Acoustics and Sound," McGraw-Hill, 1936.

26. Nichols, B.D. and C.W. Hirt, "Calculating Three-Dimensional Free Surface Flows...," J. Comp. Phys. 12 (1973), 234-46. (See also their earlier article, ibid. 8 (1971), 434-48).

27. Orszag, Steven A. and Moshe Israeli, "Numerical Simulation of Viscous Incompressible Flows," Annual Review Fluid Mech. 6 (1974), 281-318.

28. Richtmyer, R.D. and K.W. Morton, "Difference Methods for Initial-Value Problems," Wiley-Interscience, 1967.

29. Roache, Patrick J., "Computational Fluid Dynamics," Hermosa Publ., Albuquerque, N.M., 1972.

30. Schlichting, H., "Boundary Layer Theory," McGraw-Hill, 1955.

31. Sedov, L.I. and G.Y. Stepanov, "Non-Steady Flow of Water at High Speeds," Proc. IUTAM Symposium held in Leningrad, 1971, Nauka, Moscow, 1973.

32. Viecelli, J.A., "A Computing Method for Incompressible Flows Bounded by Moving Walls," J. Comp. Phys. 8 (1971), 119-43.

33. von Neumann, John, "Collected Works," 6 Vols., Pergamon, 1961.

34. Proceedings IBM Scientific Computing Symposium, "Large-Scale Problems in Physics," IBM Data Processing Div., White Plains, New York, 1965.

35. Proceedings Symp. Appl. Math., Vols. I-XX, Am. Math. Soc., 1948-68.

36. SIAM-AMS Proceedings, Vol. II, "Numerical Solution of Field Problems in Continuum Physics," Am. Math. Soc., 1970.

37. Bauer, F., P. Garabedian and D. Korn, "Supercritical Wing Sections," Springer, 1972.

APPENDIX A

A HIGHER-ORDER APPROXIMATION TO THE POISSON EQUATION

1. New Scheme. There is proposed below a new family of "finite elements" for interpolating and approximating to solutions of equations of the form

$$-\nabla^2 u + k^2 u = f(x,y), \tag{1}$$

including the Poisson and reduced wave equations (for fixed k). The resulting approximation scheme may be competitive with bicubic spline approximation for accuracy and efficiency; moreover, it has the advantage of being local, and hence applicable to rectangular polygons which are not simple rectangles.

Namely, let $u(x,y)$ be a solution of (1), e.g. a harmonic function (the special case $k = f \equiv 0$). Suppose that u is defined in a rectangular polygon P that has been subdivided into rectangles by a rectangular mesh π. At each node (s_j, u_k), consider the values of

$$u,\ v = u_{xx},\ \text{and}\ u_{yy} = k^2u - v - f(x_j,y_k). \tag{2}$$

If u is unknown, these values involve two free parameters at each node, u and v.

Given u_{jk} and v_{jk} at the meshpoints, one can interpolate a piecewise polynomial function satisfying (2) at these points, as follows. Along each edge segment, say the horizontal segment from (x_{j-1}, y_k) to $(x_j k\ y_k)$, one can first interpolate a unique cubic polynomial. This follows from general theorems of G.D. Birkhoff and Pólya.* In each mesh rectangle,

*G.D. Birkhoff, Trans. Am. Math. Soc. 7 (1906), 107-36; George Pólya, Zeits. ang. Math. Mech. 11 (1931), 445-9.

linear blending* interpolates between these cubic polynomial values along the four edges, continuous at the corners, a unique quartic and bicubic polynomial interpolant of the form

$$a_0 + a_1 x + a_2 y + a_3 x y + a_4 x^2 + a_5 x y + a_6 y^2 + a_7 x^2$$
$$+ a_8 x^2 y + a_9 x y^2 + a_{10} y^3 + a_{11} x^3 y + a_{12} x y^3.$$

This is like an Adini element, but it interpolates to the corner values of u, u_{xx}, and u_{yy} instead of to those of u, u_x, u_y.

2. Error Bounds. Along any edge between two adjacent meshpoints, the error can be bounded by a variant of G.D. Birkhoff's generalized Mean Value Theorem (op. cit.), as follows. We can normalize to the interval [-1,1], and observe that the interpolation error $e(x)$ satisfies $e(-1) = s''(-1) = e(1) = e''(1) = 0$. The Green's function for interpolation along any normalized edge is, on $\xi \leq x \leq 1$:

$$G(x,\xi) = \frac{1}{12}(1 - x)(\xi + 1)[\xi^2 - x^2 + 4\xi + 2_x + 2] \tag{3}$$

Since $u^{iv}(x) = 0$ if and only if u is a cubic polynomial, it follows that a given function and its edgewise interpolant have the same edgewise fourth derivative. As a result, a function $u(x,y)$ that satisfies $|u_{xxxx}| \leq M$ and $|u_{yyyy}| \leq M$ on edges must also satisfy on (x,y_k)

$$|e(x,y_k)| < \frac{Mh^4}{16} \int_{-1}^{1} |G([x - x_j]/h,\ \xi)|\ d\,\xi, \tag{4}$$

for all $x \in [x_{j-1}, x_j]$.

For harmonic functions, which satisy $u_{xxxx} = u_{yyyy}$, the preceding error bound along edges leads to a similar bound in the interior.

*See G. Birkhoff, J.C. Cavendish and W.J. Gordon, Proc. Nat. Acad. Sci. 71 (1974), 3423-5.

To derive this error bound, we consider first the difference u - H between the given harmonic function u, and the harmonic interpolant H(x,y) in the rectangle $R_{j,k} = [x_{j-1}, x_j]\times[y_{k-1}, y_k]$ to the cubic values interpolated along each of four edges to the nodal values of u, u_{xx} and u_{yy}. By the Maximum Principle for harmonic functions, since u - H is harmonic, the inequality (4) applies to it in $F_{j,k}$.

Finally, we consider the difference H - U between the harmonic interpolant H to the cubic polynomial edge values, and the linearly blended interpolant U constructed above. It is known* that H - U is expressible as an integral

$$\iint_{R_{j,k}} G(x,\xi)\, G(y,\eta)\, H_{xxyy}(\xi,\eta)\, d\xi\, d\eta, \tag{5}$$

since it vanished identically on $\partial R_{j,k}$. Since $H_{xxyy} = - H_{xxxx} = - H_{yyyy}$ is bounded by its edge values (being harmonic again by the Maximum Principle), we get a bound on H - U in terms of $|u_{xxxx}|_{max} = |u_{yyyy}|_{max}$]

Finally, adding to the bound on u - H and using the triangle inequality, we get

$$\begin{aligned} |u - U| &\leqq |u - H| + |H - U| \\ &\leqq K||u_{xxxx}||_{max} = K||u_{yyyy}||_{max} \end{aligned} \tag{6}$$

*G. Birkhoff and W.J. Gordon, J. Approx. Theory 1 (1968), 199-208; also G. Birkhoff, J. Cavendish and W.J. Gordon, op. cit. supra.

DISCUSSION

G. Moretti

Professor Birkhoff, I wonder if you would clarify your comment about there being no success in the numerical treatment of shock waves.

G. Birkhoff

I do not intend to imply that there has been no success. What I would like to have is a program -- a deck of cards -- which could be run through a standard computer and, for arbitrary initial conditions, really fulfill the ideas of Reimann in one dimension, but both directions. If a program exists that will reliably predict the interaction of two shock waves then it was worth coming here to know about it.

G. Moretti

I will be glad to send you a copy.

T. Sarpkaya

Going back to your mention of finite elements, I wonder if you could comment on the question of how we go about dealing with the free surface problem.

G. Birkhoff

Yes. I think that is an excellent question. I would characterize the adaptation of finite elements to free surface problems as something like the marker-and-cell program where you trace the history of motion. That is, you use geometrical elements and you also keep special track of the free surfaces. It is well known, of course, that you have to have special elements on the surface, and there are many prescriptions for treating irregular boundary elements. I have had no personal computational experience with these so I am hesitant to pass judgment. But, I see no reason to believe that you cannot get good accuracy with ingeneously designed surface elements. That, however, is research for the future

which requires fundamental inquiry into things that are obviously missing in different areas of application. The research will clearly be more difficult in three dimensions than in two.

Many people have reached the conclusion that triangular elements are preferable. I am not convinced that this is correct for fluid dynamics, especially where the Box Equation is concerned. Perhaps the adaptation of these elements to splines is worth considering. Also, a mathematician acquaintenance of mine has looked into the design of tetrahedral elements that would be sufficient for elastic compatibility, and have continuous first derivative radiance. Required are 118 unknowns per element. Although these unknowns are shared between adjacent elements, a large number of tetrahedra with complex relationships is certainly indicated. But, this is a profitable area of investigation between now and, say, 1980.

I might also mention some recent work of my own that could be useful. This deals with functions that you fit smoothly along edges by cubic splines, (piecewise cubic functions of Class C^2) or what not. This is combined with a method of linear blending discussed by many people and actually used by draftsmen, though in a non-mathematical form. This allows you to preserve the order of accuracy without greatly increasing the number of independent variables. To illustrate by the simple case of a single rectangle, what you do is to take the four corner values, interpolate a bilinear function, then interpolate linearly between the graphs of the remainder $Z = F(X,Y)$ along opposite edges. The result is a remarkably good fit. This is what draftsmen do to fair surfaces between two networks of curves, and it preserves the order of accuracy of the cubic approximation. I believe this technique should be explored in the realm of the Laplace equation.

T. Taylor

A couple of comments seem to be called for here. One is that it is nice to have a potential equation because you can formulate a variational equivalent. But, what principle do you use to formulate finite elements

for the Navier-Stokes equation? Therein lies the principle problem, and I believe Dr. Hirt will mention this in discussing some of the methodology they have developed at Los Alamos.

The second comment is simply to make it clear to the audience that a number of people have techniques to calculate flows through shock waves. Professor Moretti, obviously, is one.

J. P. Boris

I might point out that at Livermore and NRL there has been some experience with triangular systems -- combining the advantages of the triangular system having finite elements with finite difference simplicity to get rid of some of the complexities in the finite element approach.

NUMERICAL HYDRODYNAMICS -- PRESENT AND POTENTIAL

C. W. Hirt
University of California
Los Alamos Scientific Laboratory
Los Alamos, New Mexico

INTRODUCTION

Numerical techniques for the investigation of time dependent, incompressible fluid flows have been under intensive development since 1963. Time dependent flows involving free surfaces have been numerically investigated since 1965 (see Harlow 1969[10] for a partial bibliography). The Office of Naval Research has supported work in this area at the Los Alamos Scientific Laboratory since 1971.

Through the application of numerical techniques to a wide variety of problems, it has been demonstrated that numerical solutions supply an important link between experiment and pure analysis. In many cases, numerical solutions have also supplied new insight and understanding of specific flow processes. A brief review of the current state of the art clearly shows that we are now in a position to attempt some important flow problems associated with the design and performance of surface and undersea craft. While existing numerical capabilities can by no means cope with all aspects of the ship problem, there are some situations where progress can be made immediately. In addition, there are definite directions in which extensions of existing techniques can be readily made to increase the capability in this area. It should not be expected, however, that all problems will be solved, nor will all tools be available to solve them in the near future.

This paper is a review of the types of numerical techniques available for the solution of the full time-dependent, nonlinear Navier-Stokes

equations and of some calculations that have been made, which relate to ship design and performance. The examples chosen for presentation here are by no means representative of all the excellent work that has gone on in this field, as a comprehensive survey is beyond the scope of this paper. The examples chosen represent the interest and experience of the author.

Throughout the review there are suggestions for areas of research where rapid progress could be made. Some speculation is also directed to what could be accomplished in the next several years if an imaginative and active research program is undertaken.

It is hoped that the material presented here will introduce more investigators to the types of computational tools now available, and thus encourage their use and extension for more detailed studies of problems encountered in all areas of naval research.

THE COMPUTATIONAL TOOLS

One of the earliest time dependent, incompressible flow calculations to attract wide spread attention was the development of a von Karman vortex street behind a flat plate, Fromm (1963).[7] The striking similarity between Fromm's computed streaklines and Thom's experimentally obtained smoke tail, Figure 1, has caught the eye of many investigators and has stimulated widespread interest in numerical fluid dynamics. The finite difference technique employed by Fromm relied on a vorticity and stream function representation of the flow and required the solution of a Poisson equation to advance the stream function each increment in time. Use of a stream function is particularly advantageous for two-dimensional problems because it automatically insures fluid incompressibility, but it is not a convenient formulation for three-dimensional problems or for problems involving free surfaces.

To overcome some of the difficulties of the vorticity-stream function approach, Harlow and his associates (1965a)[8] developed the

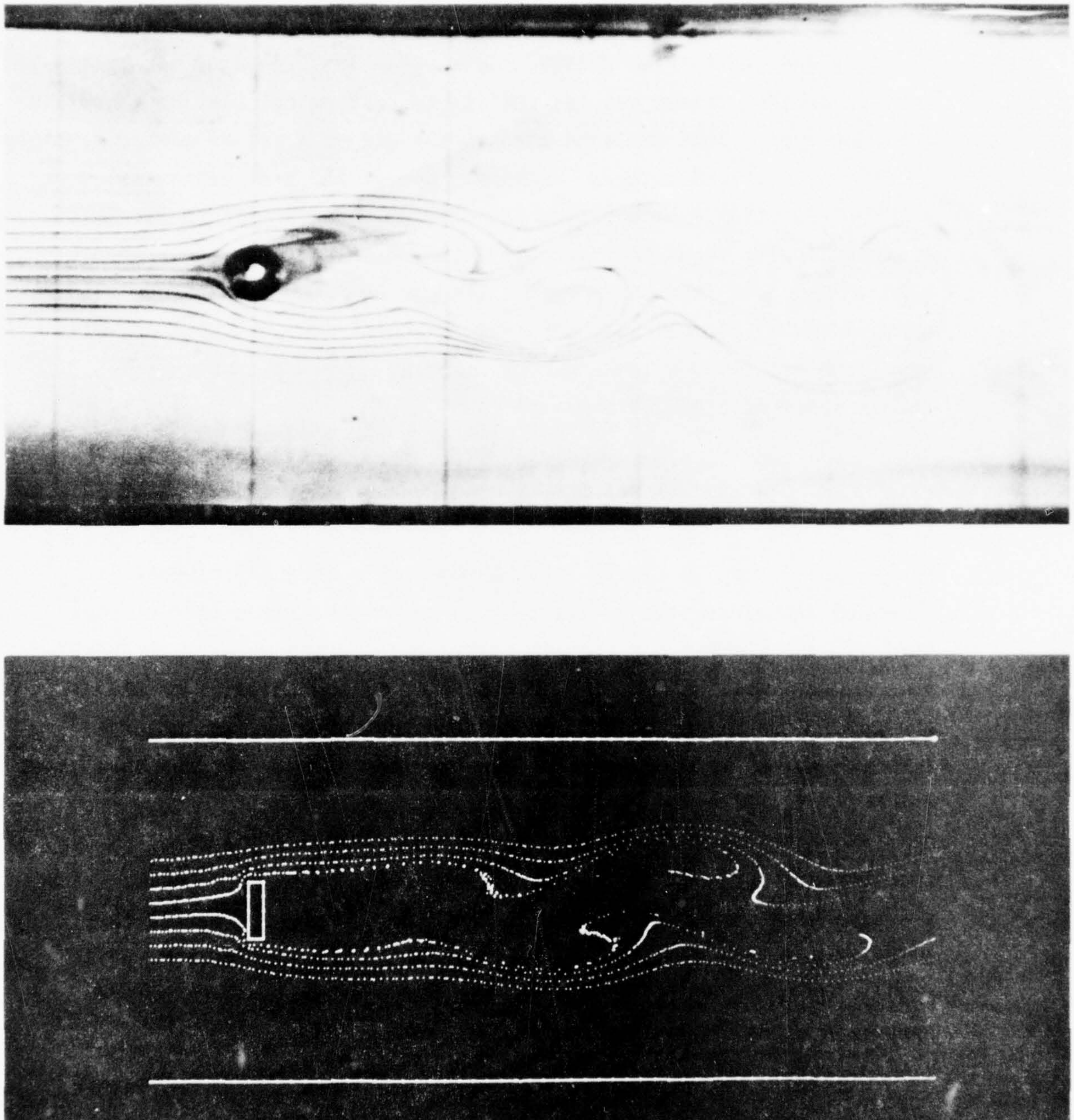


Figure 1. Comparison of Thom's Experimentally Obtained Vortex Street Behind an Obstacle at Reynolds Number 100 (Top) with Fromm's Calculation (Bottom).

Marker-and-Cell (MAC) technique, which utilizes directly the fluid velocity and pressure as primary field variables. A major accomplishment of the MAC method was its ability to perform calculations involving free surfaces. This was done through the use of a set of marker particles to define the fluid occupied regions. One of the earliest examples of a MAC calculation is the broken dam problem shown in Figure 2. This example clearly shows that strongly time-dependent, nonlinear problems with complicated free surface profiles can be readily computed. Other examples showing comparisons with experimental and analytical results have been described elsewhere, for example, Harlow and Welch (1965b),[9] Chan and Street (1970),[2] Viecelli (1969, 1971).[26]

Even some problems involving flows that should be turbulent have been computed successfully by the MAC method, provided success is measured in terms of mean flow properties. For example, the flow under a sluice gate shown in Figure 3 looks remarkably like experimentally obtained photographs and predicts a surge wave with the correct amplitude and speed. Another example is the bore formed when fluid runs into a rigid wall as depicted in Figure 4. Since these calculations are purely two-dimensional the eddy structure observed is not true turbulence, but it does produce much the same effect. In the bore case, for example, the large eddies absorb just enough kinetic energy from the mean flow to give the correct jump conditions across the bore. Since these conditions are governed by the conservation of mass and momentum, and since the numerical program is based on these conservation laws with respect to a set of small cells covering the flow region, it is really not too surprising to find such agreement between calculated and experimental results.

Several important extensions have been added to the MAC method since its introduction. Chan (1970)[2] demonstrated that the original free surface treatment in the MAC method was too crude for the detailed

study of surface waves dominated by gravity forces, because the normal stress condition was not imposed at the correct free surface location. He developed a correction that was later extended by Nichols and Hirt (1973)[21] to general surface configurations. For example, Figure 5 illustrates the complicated flow arising from the splash of a liquid drop into a pool of water.

Another limitation of the original MAC method was that it could not compute the flow about two-dimensional bodies unless their shapes coincided with the rectangular cells in the finite difference mesh. For many problems, however, nonrectangular shapes are needed. To solve this problem, Viecelli (1969, 1971)[26] developed a technique for defining bodies with curved boundaries. His method, which is a variant of the MAC free surface treatment, also permitted flows to cavitate when pressures at the solid boundaries were below a prescribed vapor pressure. Although Viecelli's method was restricted to free slip boundaries, this restriction can be removed.

Daly (1967, 1969)[4,5] has developed and applied a two-fluid variant of the MAC method and an accurate technique for the incorporation of surface tension effects.

Extensions of the MAC method for fully three-dimensional, time-dependent flows have been made by Hirt and Cook (1972),[15] and for three-dimensional flows with free surfaces by Nichols and Hirt (1973).[21] An example of the latter technique is shown in Figure 6, which illustrates the flow over a submerged rectangular body.

To improve the numerical resolution of some free surface problems several investigators, Hirt, Cook and Butler (1970a),[13] Brennan and Whitney (1970),[1] and Chan (1973),[3] have developed finite difference solution methods for the Lagrangian equations of motion for an incompressible fluid. In these schemes the finite difference mesh moves with the fluid. The advantage of a Lagrangian mesh is that it follows

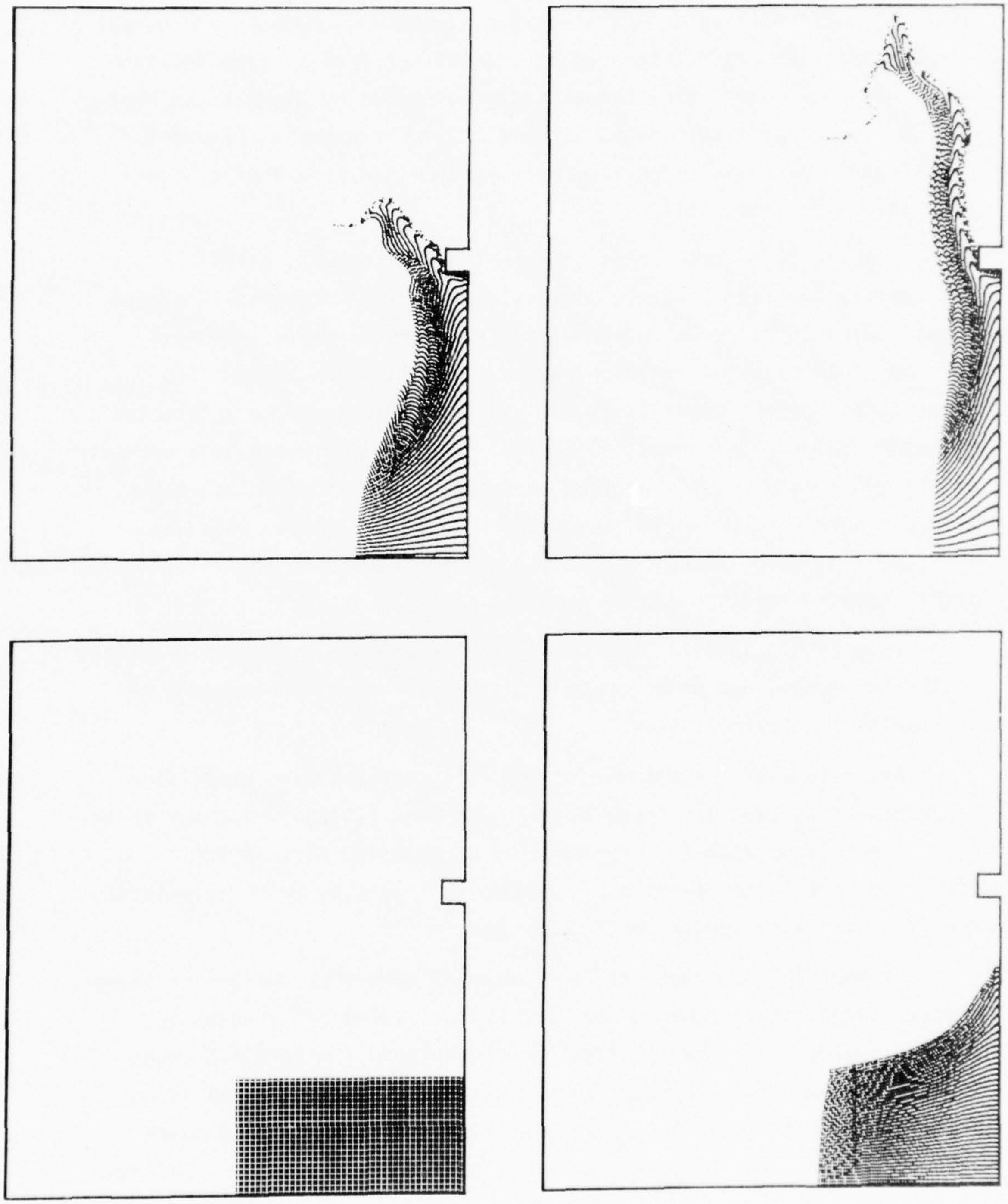


Figure 2. Collapse of a Column of Water on a Dry Bed With Splashing Over A Rigid Obstacle Down Stream

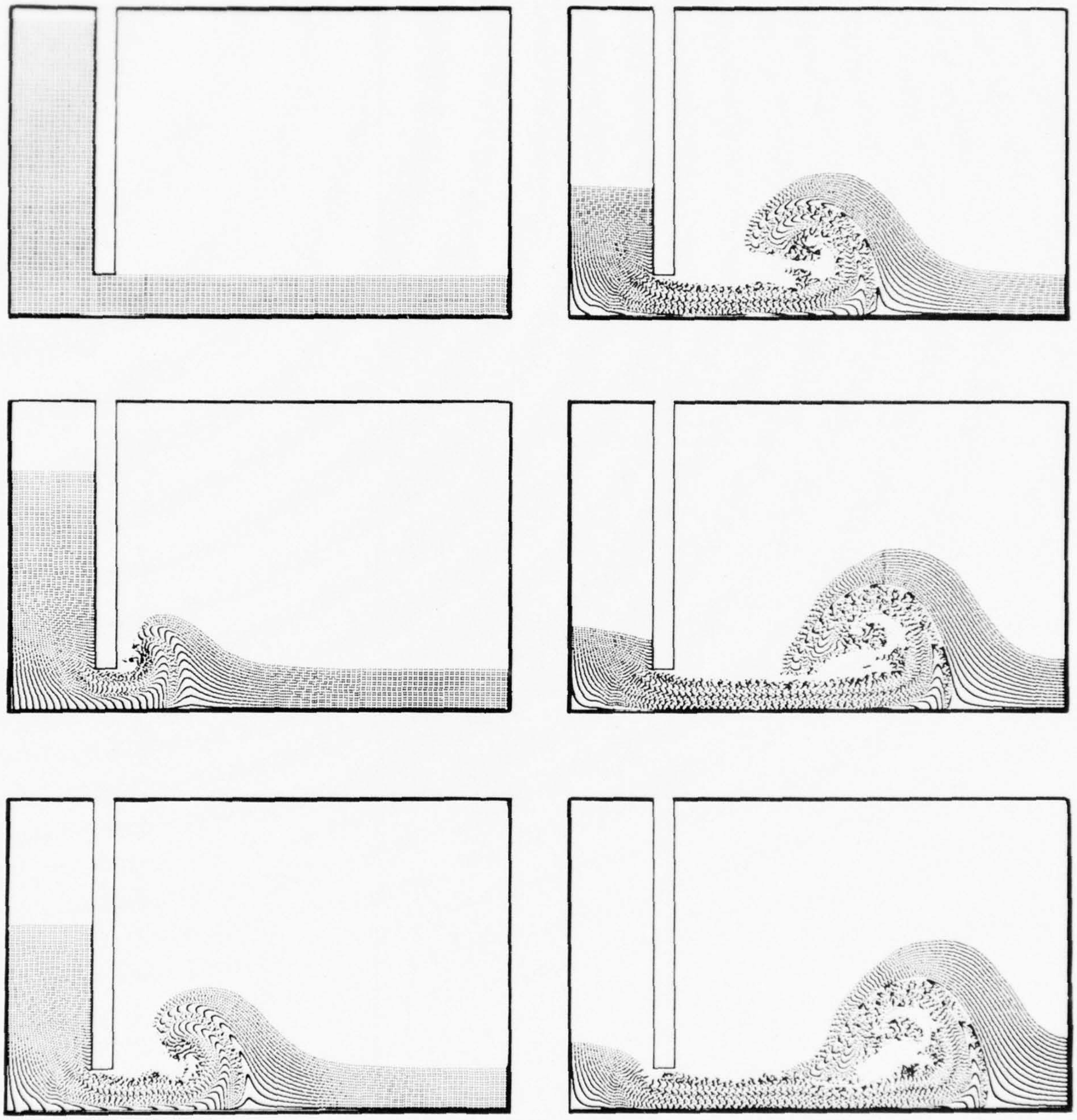


Figure 3. Flow of Water Under a Sluice Gate

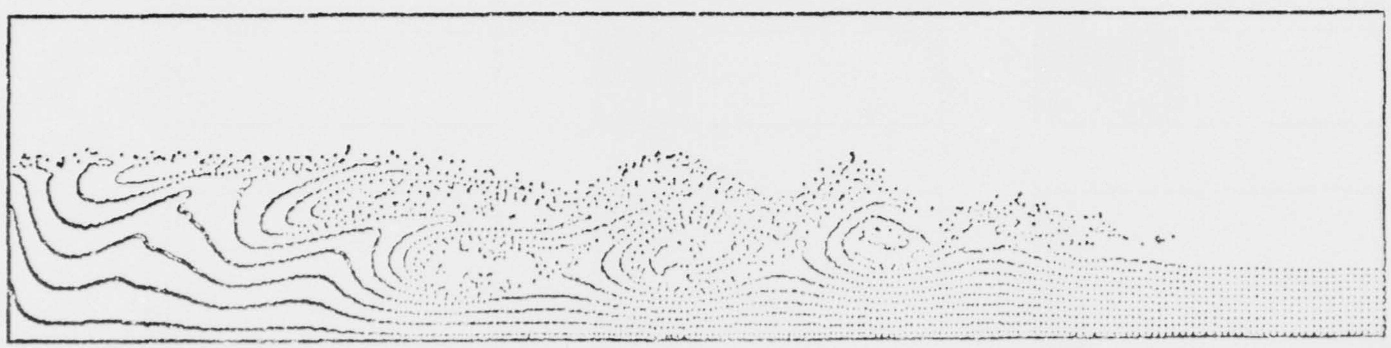


Figure 4. Structure of a Bore Formed When Fluid Coming From The Right Runs Into A Rigid Wall At The Left

Figure 5. Splash of a Liquid Drop in a Pool of Water

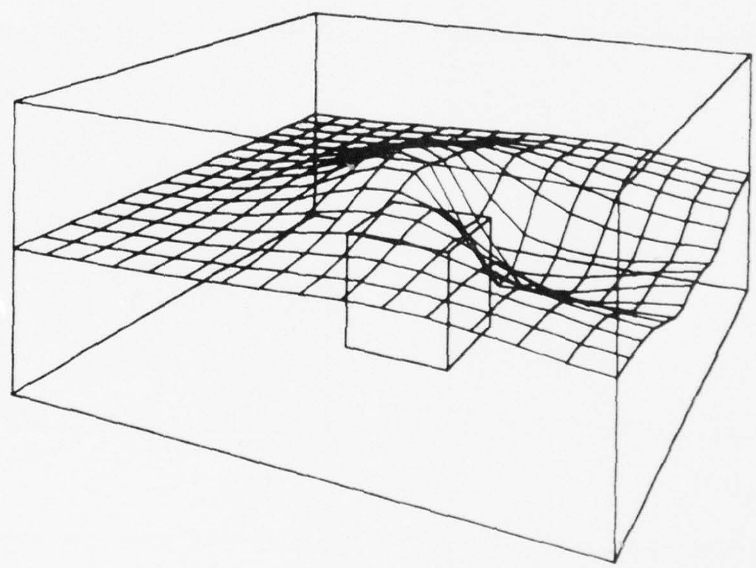


Figure 6. Three-Dimensional Flow of Water Over a Submerged Block

the free surface and other flow details and thus permits the easy application of complicated boundary conditions. Furthermore, zoning is only required in regions occupied by fluid and not in all regions where fluid is expected to go, as required in the Eulerian descriptions. It is also known, Hirt (1968),[12] that Lagrangian methods can be more accurate. An example of a Lagrangian calculation of a solitary wave reflecting from a rigid wall is shown in Figure 7.

The principle disadvantage of Lagrangian cell methods arises when large fluid deformations occur. When the mesh is too distorted, accurate calculations are no longer possible. A partial remedy for the large deformation problems was developed by Trulio (1966)[25] and by Hirt and Amsden (1970b)[14] in the form of an Arbitrary-Lagrangian-Eulerian (ALE) method. In this technique the Lagrangian mesh can be continuously rezoned to correct for excessive mesh distortions. In fact, in confined flows, the mesh can always be rezoned back to its initial configuration each time step to make it resemble a purely Eulerian computation. This feature is useful for flows about curved bodies with either free-slip or no-slip boundary conditions. For example, Figure 8 shows a flow field representing one instant in the development of pulsatile flow of blood through an artery with a constriction. The ALE method is still limited, unfortunately, when applied to flows with free surfaces, because the free surface must be treated as Lagrangian and cannot be rezoned to correct for excessive deformation.

A three-dimensional version of the ALE technique has been written by Pracht (1974),[23] but has not yet been employed for free surface problems. A three-dimensional computation of steady flow through a curved tube is illustrated in Figure 9, which shows a secondary Ekman flow.

The computational tools now available, therefore, consist of a variety of techniques for both Eulerian and Lagrangian descriptions of

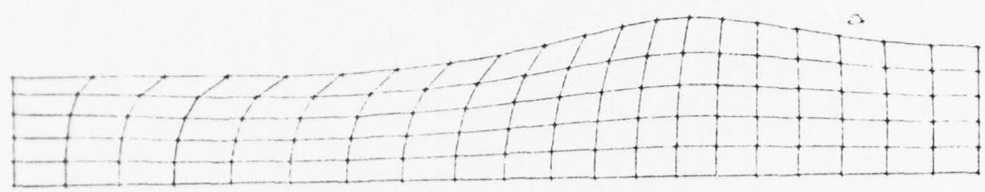


Figure 7. Solitary Wave as Computed With a Lagrangian Finite Difference Mesh

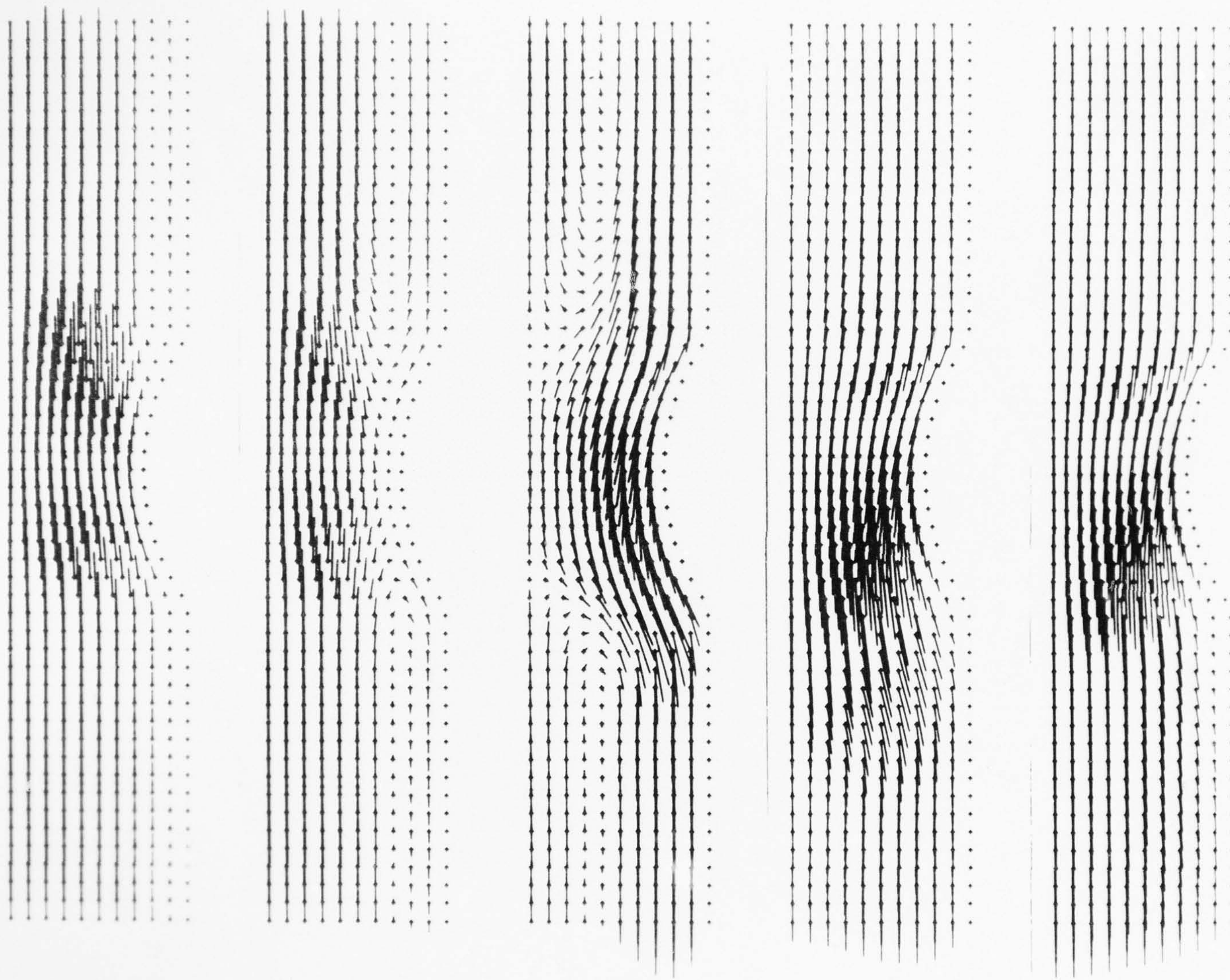


Figure 8. Velocity Fields Obtained At Different Times in a Cycle of Pulsating Blood Flow Through An Artery With a Constriction. The Left Edge of Each Frame is an Axis of Cylindrical Symmetry.

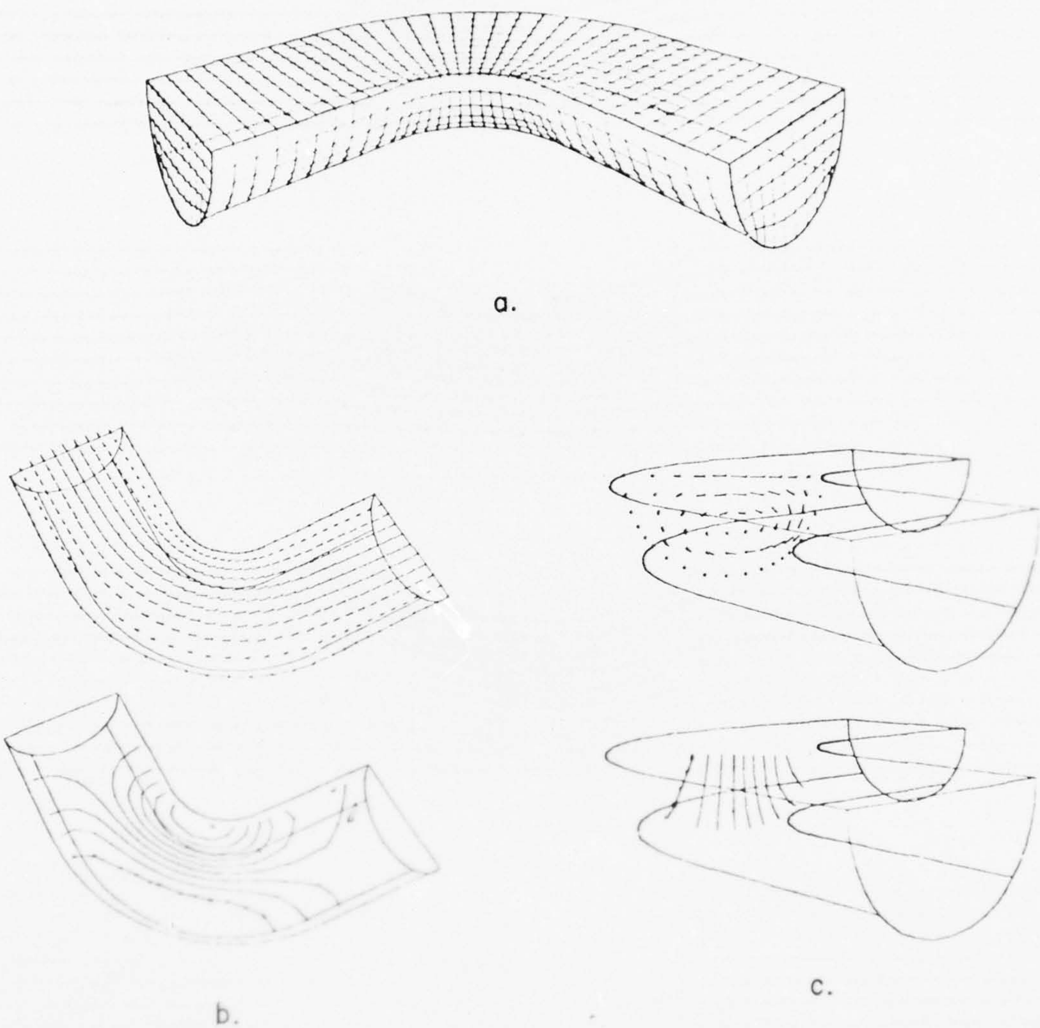


Figure 9. Three-Dimensional Flow In a Closed, Curved Tube Showing The Development of an Ekman Secondary Flow in the Bend. The Mesh is Shown in (a), Velocity and Pressure Contours in (b) and (c).

flows in two and three dimensions. Curved rigid boundaries, surface tension forces, buoyancy effects, and material interfaces are among the physical features that can be incorporated to some degree in one or more of these basic techniques. The types of problems that may be investigated with these tools are explored in the next section.

RECENT APPLICATIONS

In this section are reviewed several recent investigations involving the computation of flows about rigid bodies or flows generated by moving bodies. Many of the examples reported here were obtained with a relatively simple computer code called SOLA (Hirt, Nichols and Romero, 1974b).[17] This code contains only the essence of the Marker-and-Cell technique without the complication of special purpose features. It was written as an instructional tool to be used by persons with little or no previous experience in numerical fluid dynamics. The fact that SOLA is uncomplicated, however, has led to its use for a wide variety of applications, because major program modifications are easily incorporated with a minimum of effort. Thus, in place of a large, unwieldy code, with built-in flexibility, the SOLA code offers an alternative that is concise and easy to work with, but which must be modified for each new application.

Some idea of the range of problems to which even a simple code like SOLA can be applied will be evident from the following examples:

Wake Collapse With Shear

A problem of some current interest is the collapse of a mixed region of water embedded in a stratified ocean. The example chosen for numerical computation corresponds to a laboratory experiment performed by Wu (1969).[27] This computation was originally carried out by Young and Hirt (1972),[28] but was repeated with the SOLA code to serve as a bench mark for a study of the effects of vertical shear on wake collapse.

The computational region is 120 cm deep and 300 cm wide, and is covered by a mesh 30 cells deep and 60 cells wide. The ambient fluid is linearly stratified, which is the easiest case to set up experimentally, but any stratification may be used in the computations. The stratification chosen had a mean Väisälä frequency of 1.0 sec^{-1}. The wake region was initially circular with radius 15.6 cm, and was assumed to be uniformly mixed with no net buoyancy. The wake is located in the center of the mesh. In the example with no vertical shear the wake collapses with the subsequent radiation of internal waves that are in excellent agreement with all the experimentally obtained results (see Young and Hirt, 1972).[28] The velocity field and perturbation density contours at selected times for this computation are shown in Figures 10 and 11.

To ascertain the influence of a mean vertical shear, an effect that is difficult to include in a laboratory experiment; this calculation was repeated with a constant vertical shear, $\frac{\partial u}{\partial z}$ = constant, imposed on the ambient fluid. Two cases were considered: a very weak shear, $\frac{\partial u}{\partial z} = 0.02 \ sec^{-1}$, and a stronger shear, $\frac{\partial u}{\partial z} = 0.1 \ sec^{-1}$. The critical shear for this example, that is, the shear level where periodic waves are no longer maintained is $\frac{\partial u}{\partial z} = 2.0 \ sec^{-1}$. Thus, the strongest shear case shown is only 1/20 of the critical value, but the effect it has on the wake collapse is obviously significant, as seen in the illustrations of Figures 10 and 11. The largest shear case exhibits some Helmholtz instability in the central core region, which enhances mixing there, and at the same time the radiated disturbance remains considerably more localized.

Calculations with still larger shear values show that the wake collapse is almost entirely eliminated and density disturbances are limited to a narrow horizontal band.

Influence of Vertical Shear on Wake Collapse
VAISALA FREQUENCY = 1.0/sec

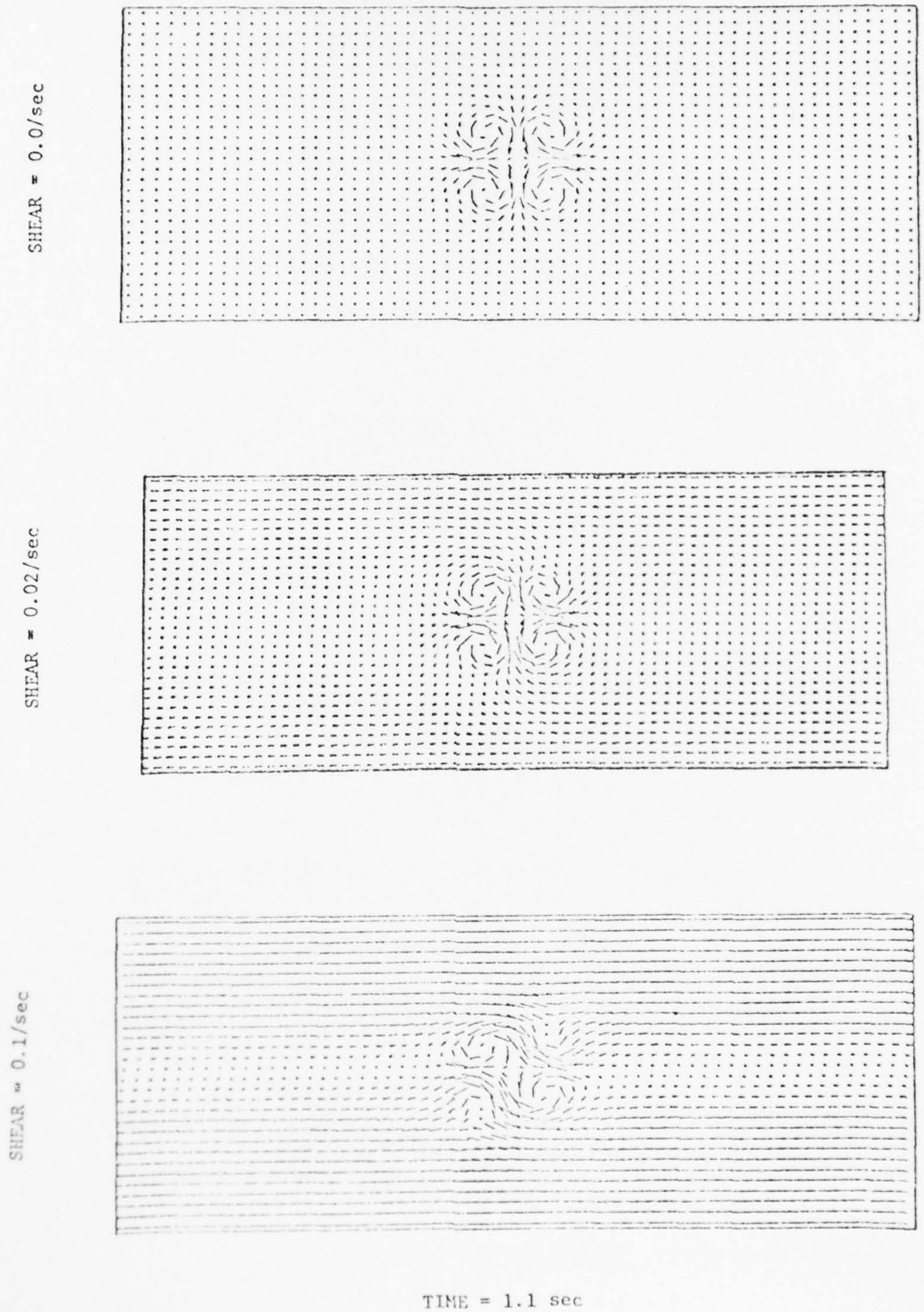
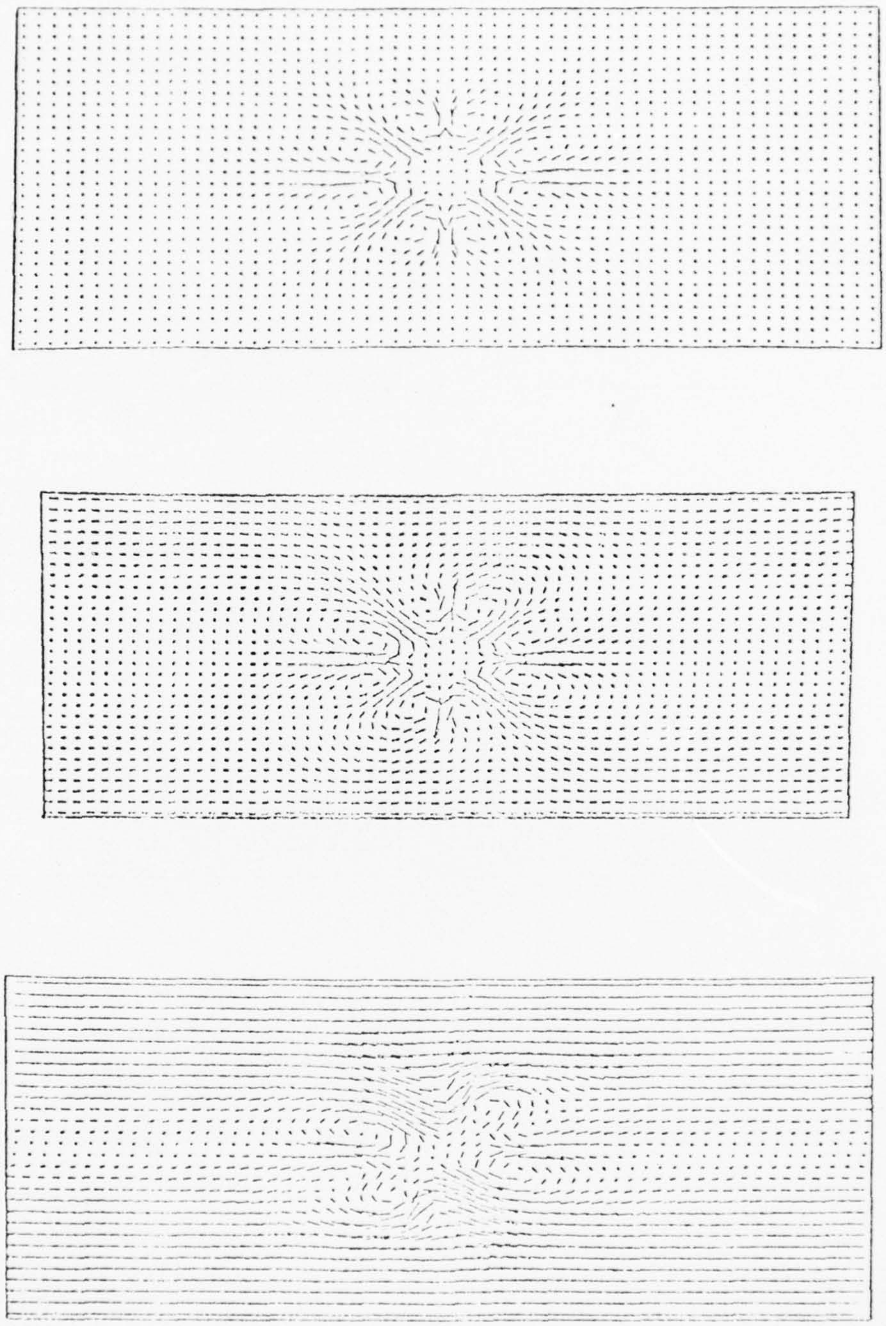
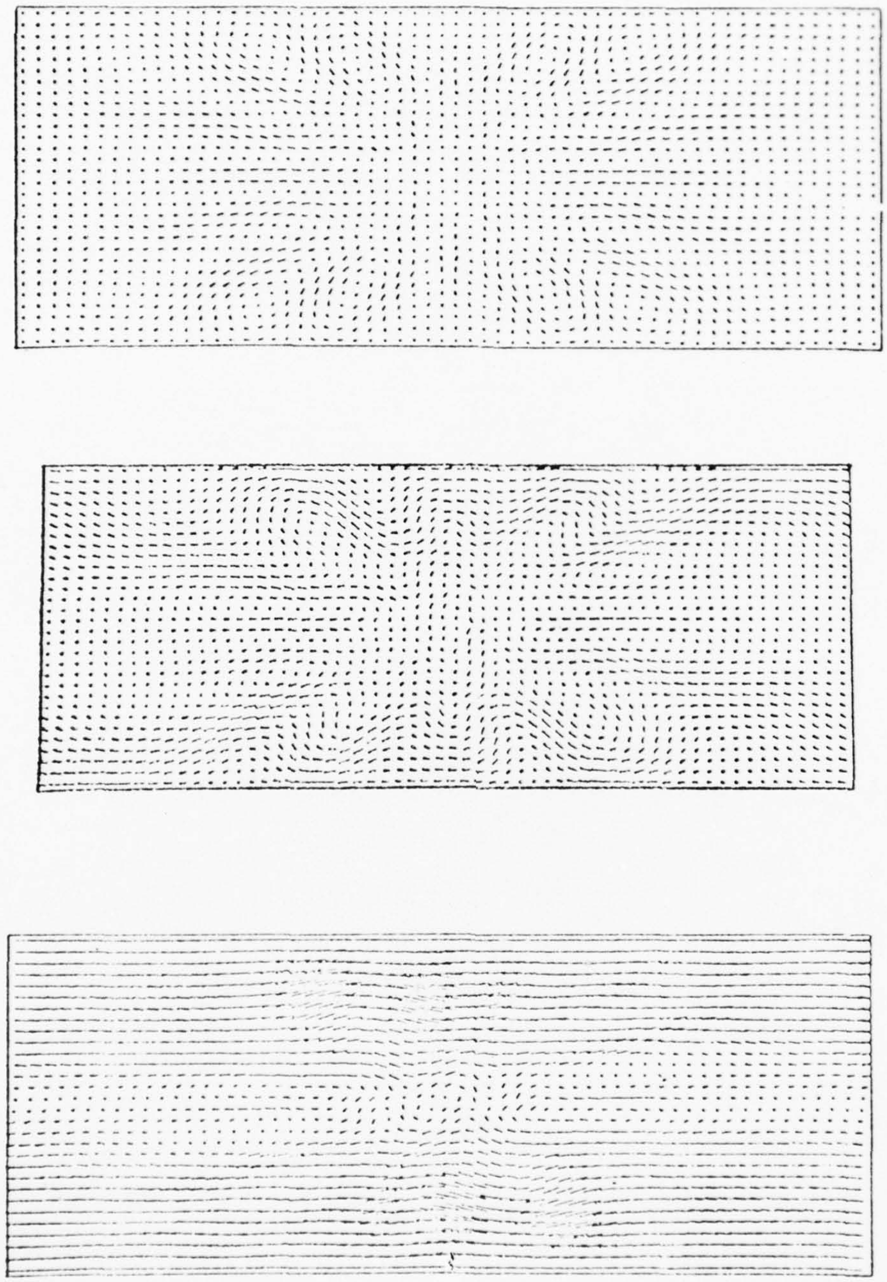


Figure 10a. Velocity Fields Caused by a Collapsing Wake In Three Different Vertical Shears at (a) 1.1 sec, (b) 5.1 sec, and (c) 14.1 sec. The Top Figure in Each Case is for No Vertical Shear, the Middle Figure Corresponds to a Shear of 0.02 Sec^{-1}, and the Bottom Figure to 0.1 Sec^{-1}.



TIME = 5.1 sec

Figure 10b.



TIME = 14.1 sec

Figure 10c.

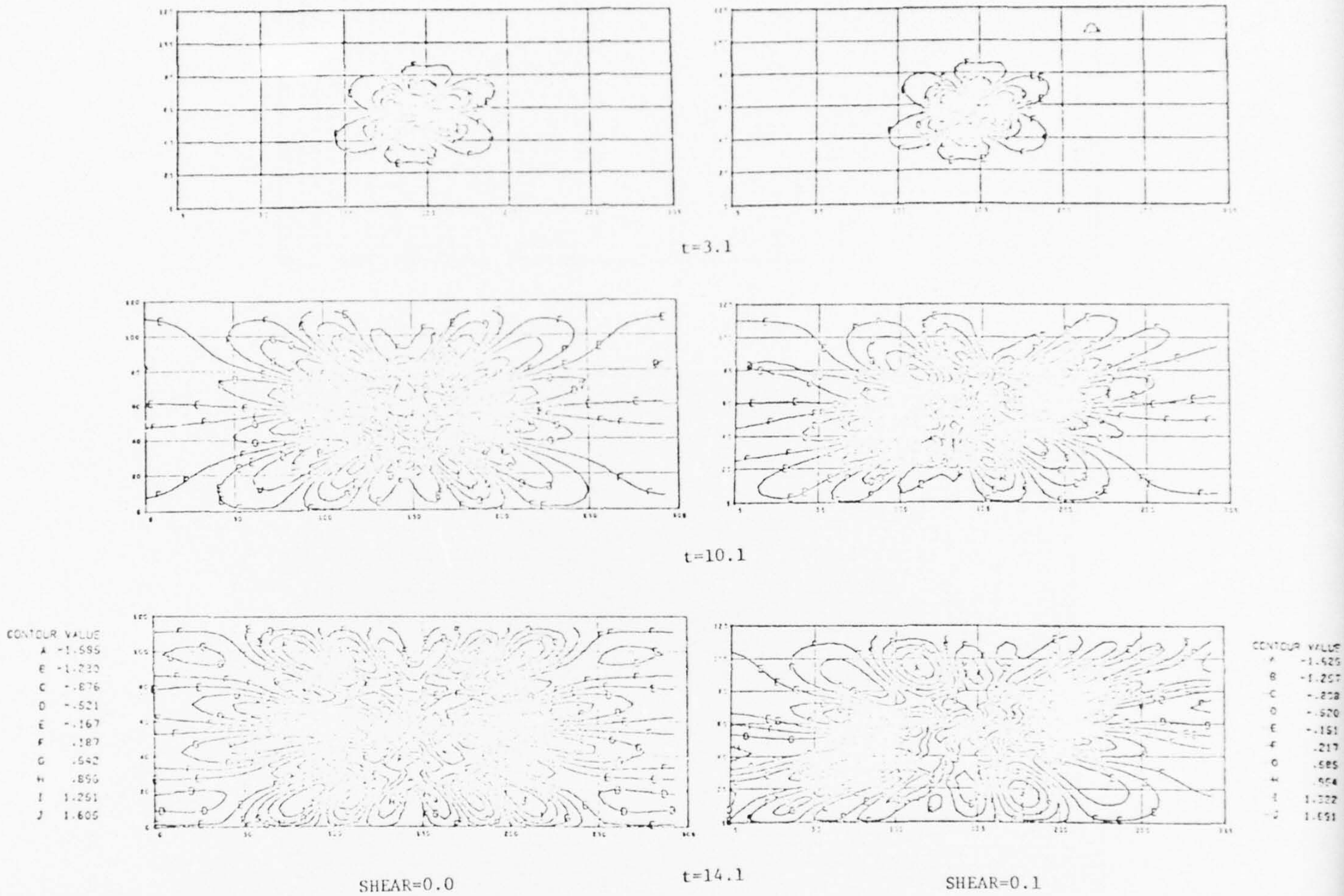


Figure 11. Perturbation Density Contours Shown at Three Times for No Shear (Left) and 0.1 Sec^{-1} Shear (Right)

Many variations in this type of calculation are possible. For example, both the density and vertical shear profiles can be arbitrary, the wake might be only partially mixed, or a net buoyancy could be given to the wake. The possibility of such variations is one reason why numerical simulations are attractive. Furthermore, each of these computations required approximately 2 or 3 minutes of CDC 7600 computer time, or roughly $20.00 per calculation, so that many runs are possible at a modest cost.

Dynamics of a Sinking Body

An interesting problem involving the sinking of an oceanographic instrument canister has been described by Munk (1973).[19] It has been observed that canisters, which are designed to sink to a neutrally buoyant depth in the ocean, sink at a rate considerably less than expected on the basis of a simple wake drag. Munk has suggested that an explanation of this might lie in the generation of internal waves or, alternatively, that bubbles of lighter fluid might be trapped in the canister's wake and buoy it up as it descends into denser fluid.

To investigate these mechanisms the SOLA code has been modified to have a cylindrical canister within its mesh and a variable density capability. Since the program is restricted to two-dimensional flows these calculations must be axisymmetric. The force on the canister was computed as it moved down into a linearly stratified fluid at a constant speed. A comparison between the calculations with and without stratification does indeed show a greater drag force tending to slow the canister in the stratified case. The wake does not form a recirculating flow of lower density fluid, however, but develops a long thin wake of lower density material, as shown in the density contours of Figure 12. Since the flow is purely axisymmetric it is not possible to have a von Karman-like wake exhibiting the periodic shedding of discrete vortices. For this reason, it was conjectured that the long

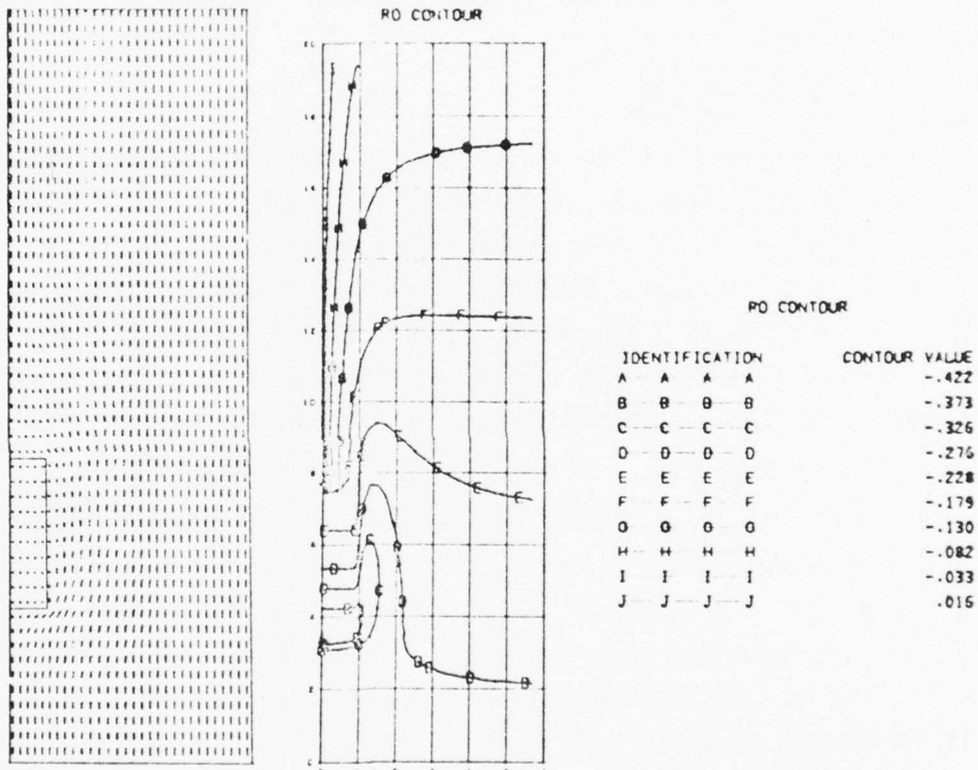


Figure 12. Velocity Field (Left) and Perturbation Density Contours (Right) Generated by a Cylindrical Can Sinking Into a Linearly Stratified Fluid. The Left Edge of Each Box is an Axis of Cylindrical Symmetry.

BEST AVAILABLE COPY

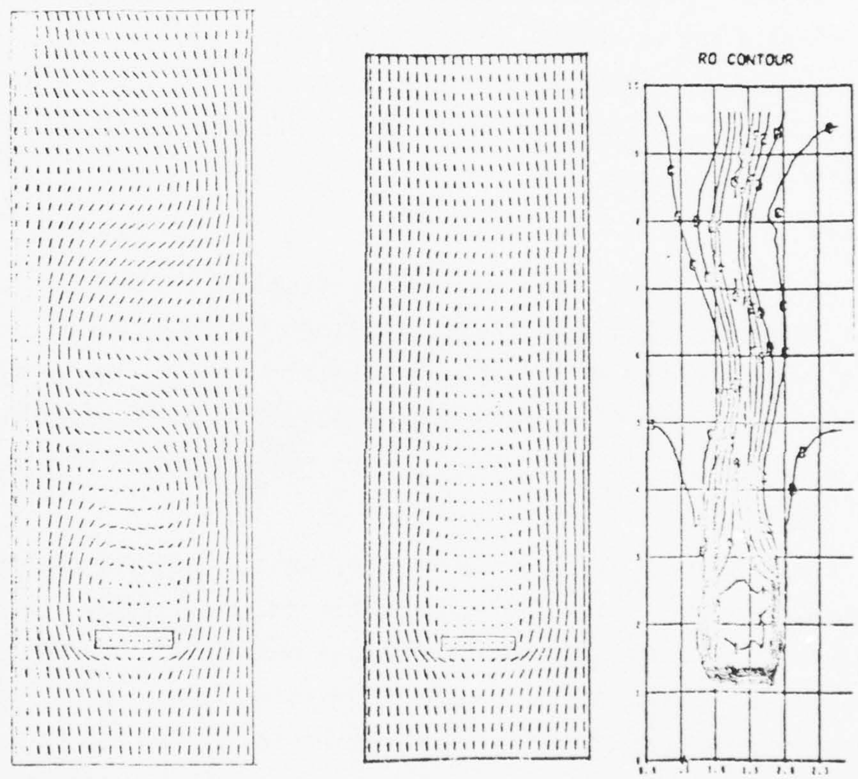


Figure 13. Velocity Vectors Showing the Flow About a Flat Plate Sinking in an Unstratified Fluid (Left) and a Linearly Stratified Fluid (Middle). Perturbation Density Contours For the Stratified Case are Shown at the Right.

wake obtained in the axisymmetric calculations might be artificial. To check this, a calculation was then performed for a rectangular plate moving through an unstratified fluid at constant speed and then another through a stratified fluid. It was found that motion in the stratified fluid resulted in a narrower wake with weak eddies shedding at a higher frequency than in the unstratified case. A comparison of these calculations is shown in Figure 13, which includes a velocity vector field for each case and a set of density contours for the stratified case. These results suggest that the long narrow wake computed for the canister may not be far wrong. Further details of this study will be reported upon its completion.

Hydrofoils

The above examples have been for confined flows, but the SOLA code can be easily modified to have a horizontal free surface, provided this surface remains single valued with respect to the vertical coordinate. More general free surface configurations are possible (see for example, Nichols and Hirt, 1971),[20] but require a more complicated computer code.

This modified version of SOLA may be used to compute the flow that develops in the vicinity of a submerged rectangular foil at zero angle of attack, Figure 14. This particular computation has not been pursued, and no attempt has been made to investigate non-zero angles of attack, non-rectangular foil shapes, or other variations, all of which would be highly interesting and are within current capabilities.

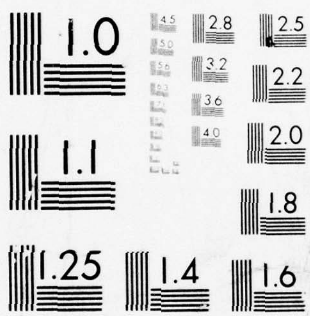
The SOLA code may also be used to compute flow over a rigid wavy surface, provided the zero pressure boundary condition used at the free surface is replaced by a zero normal velocity condition. It is further possible to employ a combination of free surface and rigid boundary conditions to study the dynamics of some two-dimensional floating bodies.

AD-A043 448 NATIONAL ACADEMY OF SCIENCES-NATIONAL RESEARCH COUNCI--ETC F/G 20/4
PROCEEDINGS OF THE WORKSHOP ON NUMERICAL HYDRODYNAMICS.(U)
JUL 75 N00014-67-A-0244-0013

UNCLASSIFIED NL

3 OF 5

AD
A043 448


1.0
4.5
5.0
2.8
2.5
3.2
2.2
3.6
4.0
2.0
1.1
1.8
1.25
1.4
1.6

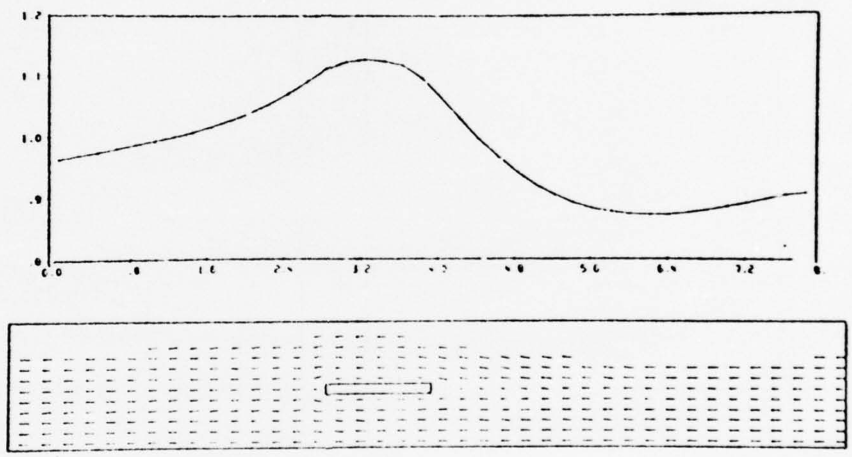


Figure 14. Flow About a Submerged Rectangular Foil; Velocity Vectors (Bottom) and Vertically Magnified Free Surface Profile (Top).

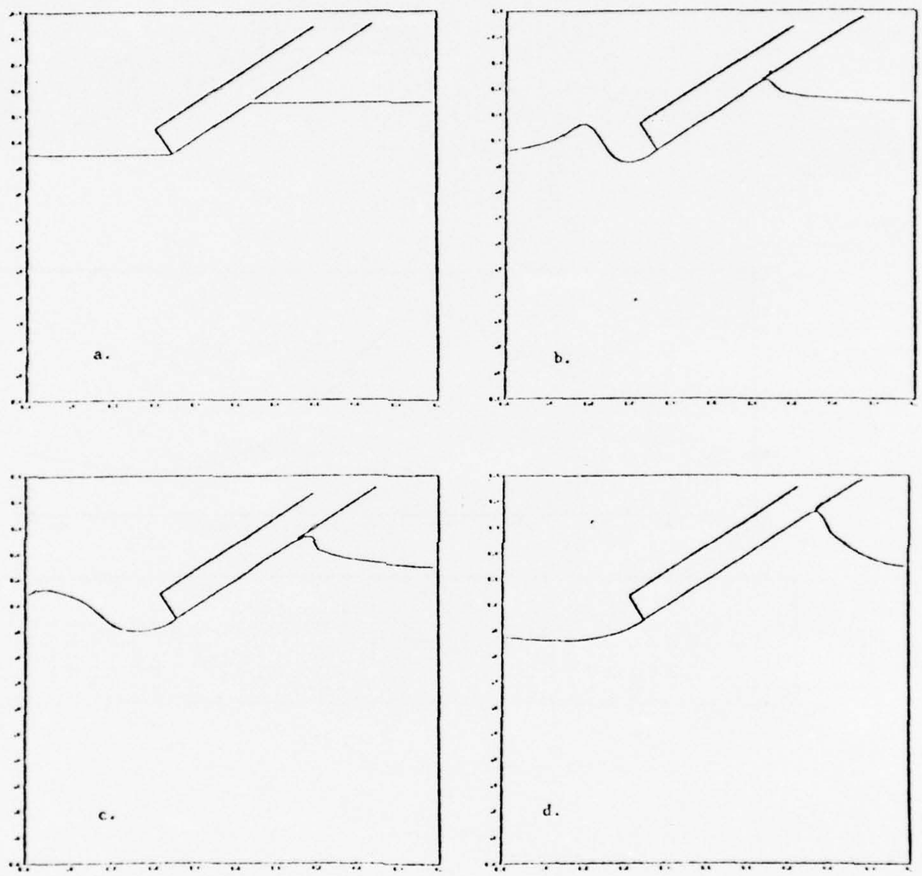


Figure 15. Free Surface Profiles at Various Times Showing the Flow Developing Around a Flat Surface Planing at a 15° Angle (Vertical Scale is Magnified Approximately Six Times More Than Horizontal Scale).

As an example of this latter technique, a computation has been made of the flow about a flat planing surface, tilted 15° above the horizontal. It is known that the fluid surface at the leading edge of the plate is not single valued, because of the formation of a forward splash. This feature was not included explicitly in the calculation for it was hoped that the numerical scheme would automatically account for the splash by permitting a loss of mass and momentum through the surface. Preliminary calculations were encouraging, as shown in Figure 15. In this example the flow is supercritical with respect to the finite fluid depth. The evolution of the surface configuration is realistic, but the pressure distribution on the plate showed considerable fluctuation, which developed because the fluid surface had a tendency to periodically bounce off the plate along its leading edge. It was believed that this bounce was not caused by the neglect of the splash, but because of the discontinuous nature of the surface boundary condition, that is, a zero surface pressure was used when the surface was free, but a large positive pressure developed when it contacted the plate, and this forced the fluid to bounce away a short time later. A continuous transition between these conditions has been developed that eliminated the difficulty. Whether or not the modified technique will now yield useful results must await a detailed study of cases where analytic and experimental data are available for comparison.

This example serves to illustrate the kind of difficulties that often arise in applying numerical solution methods to new problems. Of course, the difficulty in this example might have been eliminated by resorting to a more general numerical technique that would permit multiple valued free surfaces, higher local resolution near the leading edge, and other refinements (Chan, private communication). Our purpose in describing the simpler approach, however, is to show how simple methods are often built up in easy steps to perform difficult jobs.

The above examples show that many problems can be studied without putting a large effort into the development of a "super code." The following examples, on the other hand, illustrate situations where more sophisticated computational tools are desirable, either for reasons of accuracy or because the problems are more complex.

Floating Body Dynamics

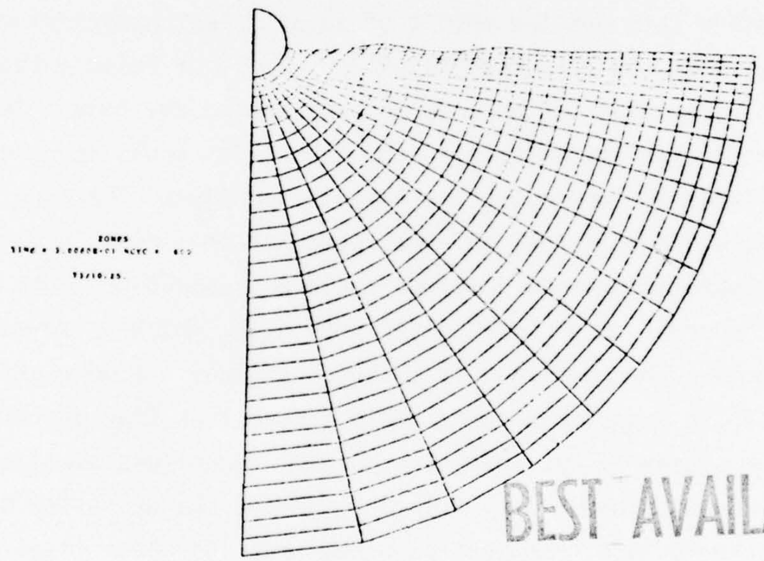
A good example where it is advantageous to choose a more sophisticated numerical technique that fits the specific needs of a problem is given in the work of Chan on the dynamics of floating bodies. Chan (1973)[3] has developed a special Lagrangian technique for the flow generated by two-dimensional floating bodies in forced heave, roll, and sway. An example of his computing mesh for the calculation of forced heave of a circular cylinder is shown in Figure 16. The Lagrangian mesh permits an accurate representation of the cylinder wall, offers good resolution near the cylinder, and permits an accurate treatment of both low and high amplitude surface waves. Through calculations such as this, Chan is able to compute virtual mass and drag coefficients for bodies of rather general shape.

As techniques like this are further developed, it will soon be possible to predict the full nonlinear dynamic response of floating bodies of arbitrary shape under a wide range of sea conditions.

Three-Dimensional Problems

Real ships are three-dimensional, and so some consideration must be given to the possibility of fully three-dimensional numerical studies. Several experimental studies of three-dimensional free surface flows have been made with a variant of the MAC method, Nichols and Hirt (1973).[21] The purpose of these calculations was threefold: to assess what could be accomplished with current technology, what the computer requirements are for meaningful calculations, and where improvements in the numerical techniques are most needed.

(a) Free Surface (Vertical Scale Has Been Exaggerated by a Factor of 50)



(b) A Part of the Computational Mesh

Flow Configuration After Six Complete Oscillations

Figure 16. Lagrangian Mesh Calculation of Cylinder in Forced Heave. Magnified Free Surface Profile Shown at Top and a Portion of the Mesh Near the Cylinder Shown Below.

The calculations performed were successful in showing that many interesting, strongly three-dimensional problems can be computed. For example, Figures 6 and 17 illustrate the surface profiles obtained in the vicinity of rectangular bodies. The wake flow shown in Figure 17 exhibited a periodic behavior with the fluid level at the rear of the body sloshing up and down and a three-dimensional eddy-like flow appearing and disappearing in the wake. This calculation employed a plane of symmetry at the wake midplane. When the calculation was repeated for the entire wake region, the wake did not exhibit an asymmetric shedding of vortices as might have been expected. It was unclear whether this was the result of insufficient numerical resolution, which implies a low effective Reynolds number, or because there was insufficient space for the wake to develop behind the body. In either case, a mesh with variable cell size capability would have helped without significantly increasing the computation time. The best situation, of course, would be to use a larger computational mesh, provided the longer computational time required could be justified. This calculation took on the order of an hour of CDC 7600 time, which is somewhat excessive when more than one case is to be considered. A more efficient version of the computer program would reduce this time approximately in half. For steady flows, there are special techniques available to accelerate the attainment of a solution without the necessity of computing all the intermediate transient flow features, so these cases are not as formidable.

The free surface treatment used in the above examples is limited to single valued profiles. An extension under development by Nichols will permit highly distorted surfaces including breaking waves and splashes. Anticipated applications of this more general approach are the detailed study of breaking bow waves and the generation of splashes during slamming. An early calculation made by Nichols is shown in Figure 18, which pictures the evolution of a bow wave generated by a blunt body that has been impulsively set into motion.

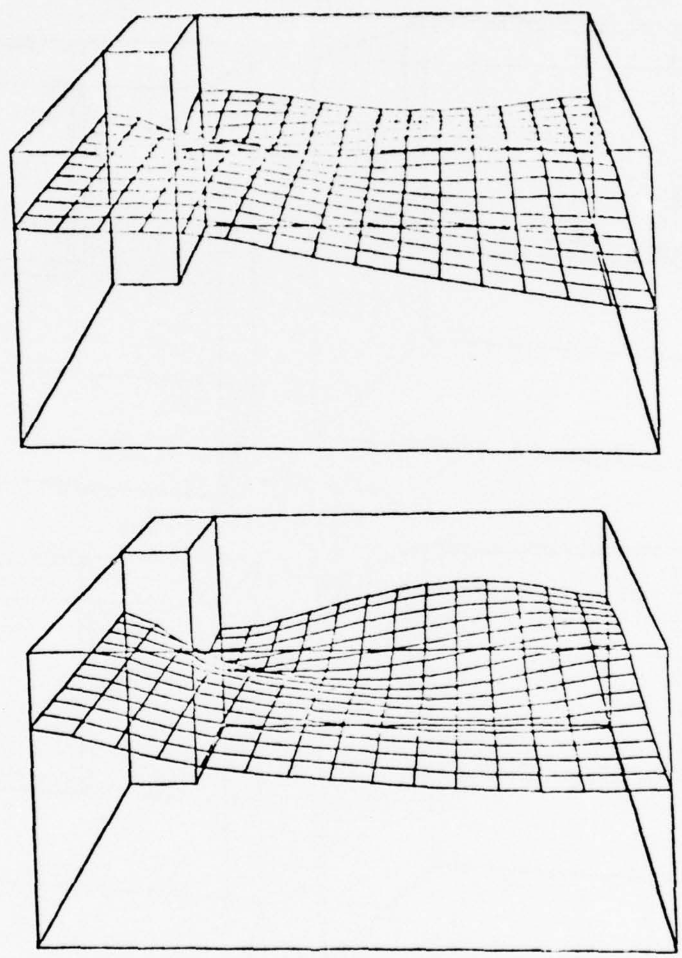


Figure 17. Three-Dimensional Calculation of Flow in the Wake of a Rectangular Body. The Wake Oscillates Periodically Between the Two Configurations Shown.

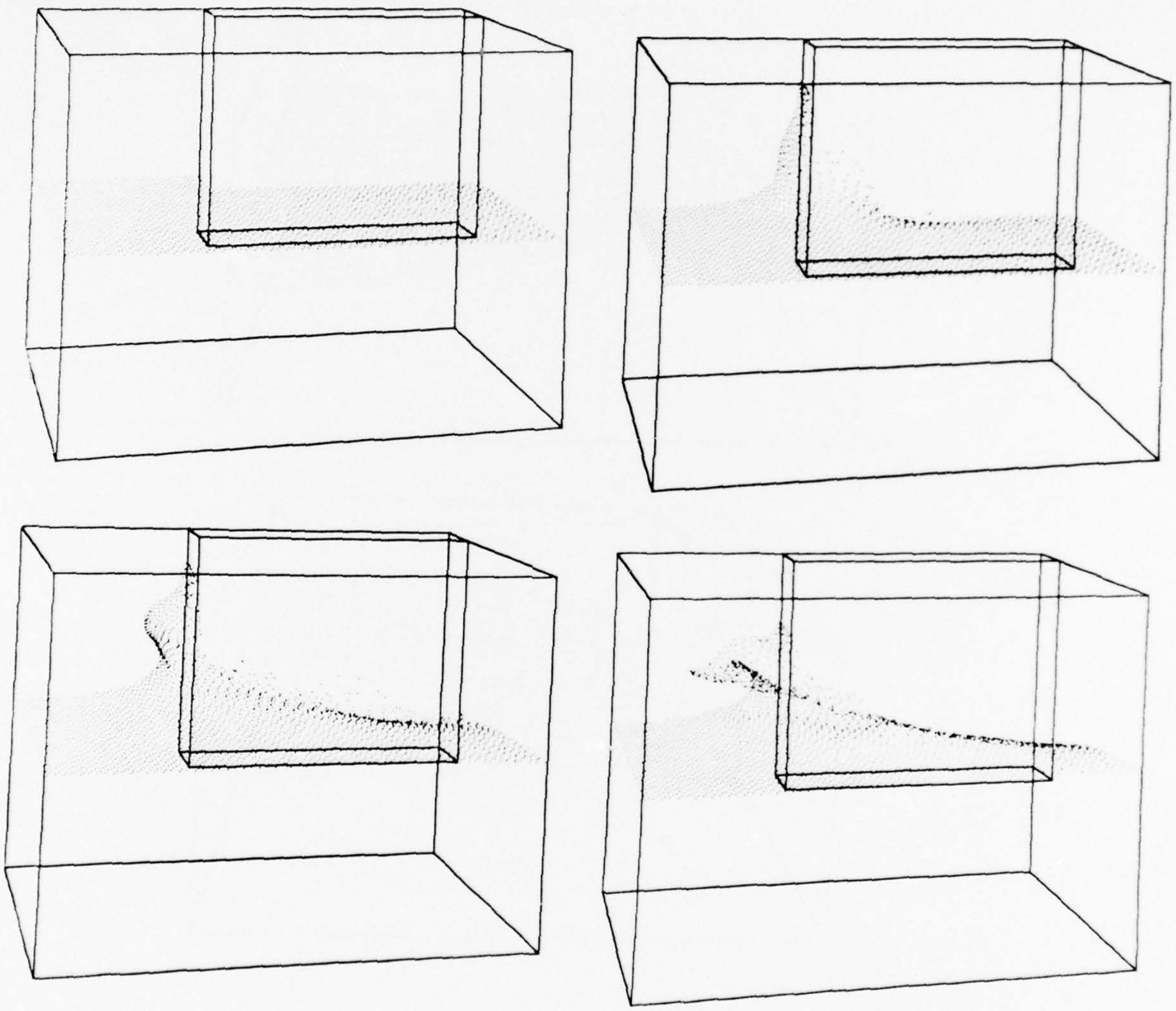


Figure 18. The Three-Dimensional Calculation of a Bow Wave Breaking In Front of a Blunt Body Set Impulsively into Motion.

This technique could also be used to investigate wave impact forces, form drag around appendages, and near wake flows for propeller design considerations.

The only real difficulty in obtaining three-dimensional numerical results is cost. The streamlining of current programs, incorporation of variable meshes, and other techniques to improve accuracy, will bring many three-dimensional problems into the range of a modest computer budget.

PRESENT LIMITATIONS AND FUTURE DEVELOPMENTS

There now exist many numerical techniques capable of resolving important problems in the design and performance of ships. The present limitations of these techniques and the kinds of new developments expected in the near future are the subject of this section. These two topics are introduced together because new techniques are likely to arise from attempts to eliminate current limitations.

The Basic Limitations

The single most serious limitation of the numerical methods described in this paper is one of resolution. The restrictions imposed by limited computer memory and time often prevent the accurate representation of fine details embedded in a larger flow region; for example, the boundary layer structure of a large ship as opposed to its wake. Alternatively, resolution of fine details often limits the flow region to local effects; for example, the near wake of a ship might be adequately computed but not the far field, radiated wave pattern.

To some extent, the problems associated with widely different spatial (and temporal) scales can be overcome through the use of variable mesh schemes. Another, potentially better, technique currently under development by Young and Ko (1973)[29] for the study of the wave field produced by a moving submerged body, consists of a numerical solution for

the near field coupled to an analytic solution for the far field. It is likely that many more combined analytic and numerical techniques of this nature will appear as better coupling schemes are devised. Finite element and Galerkin approximation methods may be one way to accomplishing this coupling in a natural way. It is also likely that other kinds of combined schemes will arise. For example, it is not hard to imagine the analytic reduction of equations into sets of simpler equations governing different space and time scales, with numerical solutions then obtained for the reduced equations. If some of the equations could be solved analytically, all the better. A method of incorporating boundary layer effects into a calculation of the wave field around a body might be developed using the idea of matched expansions, which has been used effectively in many analytic studies.

Another type of resolution problem is that associated with the errors introduced by using discrete approximations. These errors are known to produce dispersive and dissipative effects that sometimes obscure corresponding physical mechanisms. An often quoted limitation in this category arises from the occurrence of diffusion-like errors in the finite difference momentum equations, which may obscure the influence of a real viscosity and, hence, limit their application to flows with relatively low Reynolds numbers. Simple arguments can be given, Hirt and Cook (1972),[15] to show that the maximum Reynolds number accurately computed in a mesh with N cells resolving a typical linear dimension is of order N^2. Currently, N is typically of order 10 to 100.

This does not mean, however, that meaningful calculations cannot be performed at much higher Reynolds numbers. In many problems the influence of viscosity is negligible once it is reduced below a threshold level. Even when the Reynolds number of a calculation cannot be reduced sufficiently, the qualitative features of a calculation may be adequate to resolve the basic flow structure and its dependence on other

physical parameters. Sometimes the results of several calculations at different Reynolds numbers can be extrapolated to the zero viscosity limit. In some cases higher order numerical approximations, Orszag (1974),[22] or schemes involving the subtraction of known errors, Rivard, et al (1973),[24] can be used to improve accuracy. No hard and fast rules can be applied to predict, *a priori*, the influence of this type of resolution error.

When the effects of turbulence are of prime importance, recourse can be made to simple models for eddy viscosities or more complex models involving the simultaneous numerical solution of equations describing various turbulence quantities (see Daly and Harlow, 1970).[6]

Future Developments

An idea that has yet to be explored, but one that might provide a useful way to model turbulent flows, would involve the Monte Carlo treatment of turbulent fluctuations coupled to a finite difference representation of the mean flow. Monte Carlo methods might also be useful for the representation of statistical forces on floating bodies caused by random wave fields, or for the interaction of submarine generated internal waves with a random background of internal waves and localized shear flows.

A recently developed modeling technique that has proven flexible and useful for various air pollution dispersal problems, Hotchkiss and Hirt (1972),[18] employs a set of discrete particles to represent masses of pollutant material. These particles are treated as Lagrangian elements insofar as they move with the mean air flow, but are also treated statistically to represent turbulent dispersal. The particles may have inertial, gravity, and other forces acting on them. An example of the use of these particles to represent a smoke plume originating on the top of a building is shown in Figure 19. There is no reason why this scheme could not be used for the trajectory of gases from a ship's stack.

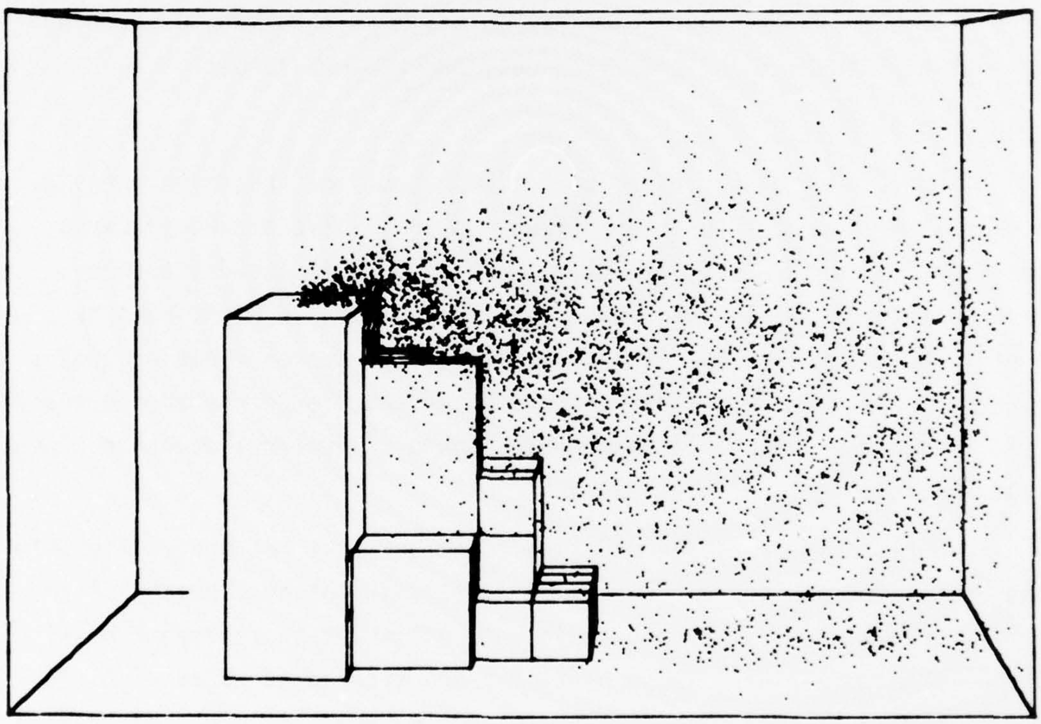


Figure 19. Dispersal and Wake Entrainment of Smoke Issuing From a Flush Vent on Top of a Building Complex.

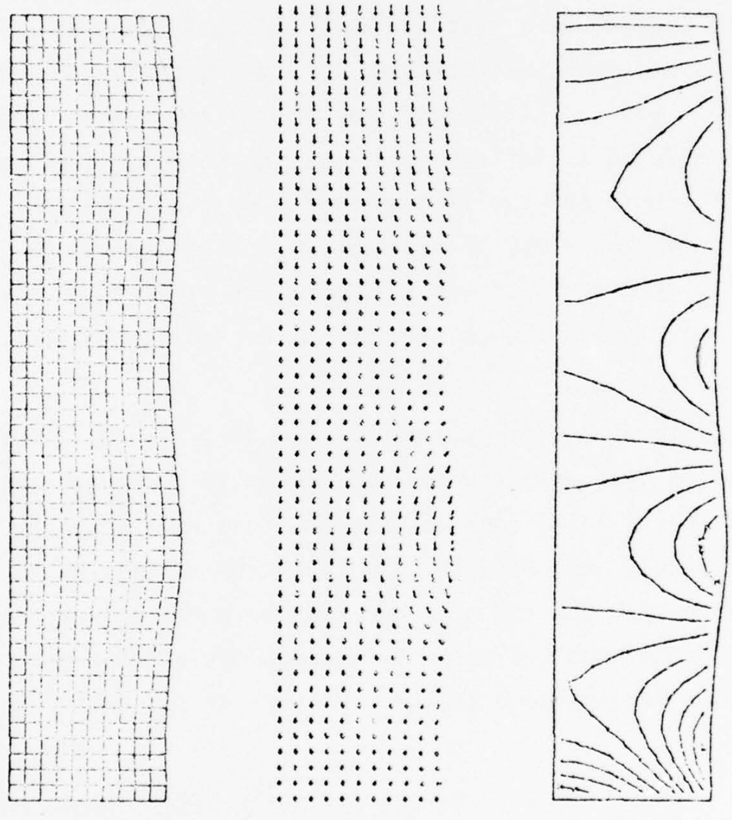


Figure 20. Pulsatile Flow in an Elastic Tube as Calculated With a Deformable Mesh Shown on Left (Axis of Symmetry is at Left Edge of Mesh). Velocity Vectors are Shown in the Center and Pressure Contours at the Right.

Other uses for this type of discrete particle model are easy to imagine. The particles could be used to represent a spray of water droplets flowing over the bow of a ship, provided there is a mechanism for their generation when a ship slams into a wave. Particles could also be used to represent small density, or acoustic, signals propagating in a nonuniform or random medium. For example, acoustic noise, i.e., density fluctuations, could be represented by sets of particles, which are created by a suitable model of turbulence or other generating mechanism, and moved with respect to the ambient fluid with the local velocity of sound. Effects of scattering, absorption, and so forth, could be modeled as influences affecting the history of each particle. A method of this nature, in conjunction with a turbulence model, would be useful for the study of hydrodynamic noise generation. This method would also be useful for investigating the scattering of acoustic signals from irregular shaped structures, or from structures with moving parts.

Particle models could also be used as the basis for a numerical treatment of sediment transport and deposition, to be employed in the design of naval shore installations. Sediment transport, however, has other difficult aspects associated with the initial entrainment of sand and with the development of a bed-load or surface creep region. Nevertheless, these problems are not insurmountable and a useful numerical model could be developed in the next year or two with a concentrated effort.

A new numerical technique currently under development for several kinds of reactor safety studies, Harlow and Amsden (1974),[11] considers the dynamics of a composite medium composed of droplets and vapor (or bubbles and liquid). It is anticipated that this technique will provide a means of investigating nonsteady cavitating flows, as well as the flow of water heavily loaded by sand or air bubbles, and the flow of air heavily loaded with water droplets.

The arbitrary Lagrangian and Eulerian (ALE) computing technique mentioned earlier could be used for the investigation of flows about elastically deforming structures. The method has already been used for a calculation of pulsatile flow in an elastic tube as shown in Figure 20. As three-dimensional methods are refined, or larger computer budgets become available, a three-dimensional ALE scheme could be used to investigate such problems as the design of non-rigid propellers.

The future looks bright. There appear to be numerous problems that would yield to concentrated numerical attacks. In a few cases the necessary tools have already been developed and are available for immediate use. Much more could be achieved by combining existing techniques together in new ways and adding various small extensions. With a continuing effort in the development of new techniques, and with the increased availability of larger and faster computers, some major inroads into presently intractable problems will assuredly be made in the next several years.

ACKNOWLEDGEMENT

Nearly all of the work described in this paper has been inspired or directly influenced by Francis Harlow. It is a pleasure to acknowledge his friendship and guidance. This paper would also not have been possible without the dedicated contributions of Billy Nichols, Nicholas Romero, and Robert K.-C. Chan.

REFERENCES

1. Brennen, C.B. and A.K. Whitney, (1970), "Unsteady, Free Surface Flows; Solutions Employing the Lagrangian Description of the Motion," Proceedings of the 8th Symposium on Naval Hydrodynamics, Pasadena, California, 1970.

2. Chan, R.K.-C. and R.L. Street, (1970), "SUMMAC - A Numerical Model for Water Waves," Department of Civil Engineering, Technical Report Number 135, Stanford University; "A Computer Study of Finite-Amplitide Water Waves," Jour. Comp. Phys., 6, 1.

3. Chan, R.K.-C., (1973), "A Generalized Arbitrary Lagrangian-Eulerian (GALE) Method for Incompressible Flows with Sharp Interfaces," Science Applications, Inc., Report Number SAI-73-575-LJ.

4. Daly, B.J., (1967), "A Numerical Study of Two-Fluid Rayleigh-Taylor Instability," Physics of Fluids, 10, 297.

5. Daly, B.J., (1969), "A Technique for Including Surface Tension Effects in Hydrodynamic Calculations," Journal Comp. Phys., 4, 97.

6. Daly, B.J. and F.H. Harlow, (1970), "Transport Equations in Turbulence," Physics of Fluids, 13, 2634.

7. Fromm, J.E., (1963), "A Method for Computing Nonsteady, Incompressible, Viscous Fluid Flows," Los Alamos Scientific Laboratory Report LA-2910; "The Time Dependent Flow of an Incompressible Viscous Fluid," in Methods of Computational Physics, B. Adler, S. Fernbach and M. Rotenberg, Eds., Academic Press, Inc., New York.

8. Harlow, F.H. and J.E. Welch, (1965a), "Numerical Calculation of Time-Dependent Viscous Incompressible Flow," Physics of Fluids, 8, 2182; Welch, J.E., F.H. Harlow, J.P. Shannon and B.J. Daly, (1965), "The MAC Method," Los Alamos Scientific Laboratory Report LA-3245.

9. Harlow, F.H. and J.E. Welch, (1965b), "Numerical Study of Large Amplitude Free Surface Motions," Physics of Fluids, 9, 842.

10. Harlow, F.H., (1969), "Numerical Methods for Fluid Dynamics, an Annotated Bibliography," Los Alamos Scientific Laboratory Report LA-4281.

11. Harlow, F.H. and A.A. Amsden, (1974), "Numerical Calculation of Multiphase Fluid Flow," in preparation for Journal Comp. Phys.

12. Hirt, C.W., (1968), "Heuristic Stability Theory for Finite Difference Equations," Journal Comp. Phys., 2, 339.

13. Hirt, C.W., J.L. Cook and T.D. Butler, (1970a), "A Lagrangian Method for Calculating the Dynamics of an Incompressible Fluid With Free Surface," Journal Comp. Phys., 5, 103.

14. Hirt, C.W., (1970b), "An Arbitrary Lagrangian-Eulerian Computing Technique," Proceedings Second International Conference on Numerical Methods in Fluid Dynamics, Berkeley, California, September 1970.

15. Hirt, C.W. and J.L. Cook, (1972), "Calculating Three-Dimensional Flows Around Structures and Over Rough Terrain," Journal Comp. Phys., 10, 324.

16. Hirt, C.W. and A.A. Amsden, (1974a), "An Arbitrary Lagrangian-Eulerian Computing Method for All Flow Speeds," Journal Comp. Phys., 14, to appear; Amsden, A.A. and C.W. Hirt, (1973), "YAQUI: An Arbitrary Lagrangian-Eulerian Computer Program for Fluid Flows at All Speeds," Los Alamos Scientific Laboratory Report LA-5100.

17. Hirt, C.W., B.D. Nichols and N.C. Romero, (1974b), "A Numerical Solution Algorithm for Incompressible Fluid Flows," Los Alamos Scientific Report, in preparation.

18. Hotchkiss, R.S. and C.W. Hirt, (1972), "Particulate Transport in Highly Distorted Three-Dimensional Flow Fields," Proceedings Computer Simulation Conference, San Diego, California, June 1972.

19. Munk, W., (1973), private communication.

20. Nichols, B.D. and C.W. Hirt, (1971), "Improved Free Surface Boundary Conditions for Numerical Incompressible-Flow Calculations," Journal Comp. Phys., 8, 434.

21. Nichols, B.D. and C.W. Hirt, (1973), "Calculating Three-Dimensional Free Surface Flows in the Vicinity of Submerged and Exposed Structures, "Journal Comp. Phys., 12, 234.

22. Orszag, S.A. and L.W. Jayne, (1974), "Local Errors of Difference Approximations to Hyperbolic Equations," Journal Comp. Phys., 14, 93.

23. Pracht, W.E., (1974), "Calculating Three-Dimensional Fluid Flows at All Speeds with An Eulerian-Lagrangian Computing Method," in preparation.

24. Rivard, W.C., O.A. Farmer, T.D. Butler and P.J. O'Rourke, (1973), "A Method for Increased Accuracy in Eulerian Fluid Dynamics Calculations," Los Alamos Scientific Laboratory Report LA-5426-MS.

25. Trulio, J.G., (1966), Air Force Weapons Laboratory, Kirtland Air Force Base, Report Number AFWL-TR-66-19.

26. Viecelli, J.A., (1969), "A Method for Including Arbitrary External Boundaries in the MAC Incompressible Fluid Computing Technique," Journal Comp. Phys., 4, 543; (1971), "A Computing Method for Incompressible Flows Bounded by Moving Walls," Journal Comp. Phys., 8, 119.

27. Wu, J., (1969), "Mixed Region Collapse with Internal Wave Generation in a Density-Stratified Medium," Journal Fluid Mech., 35, 531.

28. Young, J.A. and C.W. Hirt, (1972), "Numerical Calculations of Internal Wave Motions," Journal Fluid Mech., 56, 265.

29. Young, J.A. and D. Ko, (1973), private communication.

DISCUSSION

J. Enig

I notice that in all your examples involving rigid bodies and fluids, the rigid body is stationary and generally follows the coordinate lines, and the fluid moves past the body. Have you ever worked any problems in which the body is moving through the fluid in either a prescribed or unprescribed fashion?

C. W. Hirt

Yes, we have. In the case of the simpler program I described at the outset, when you are putting in the surface, as long as the fluid surface is single valued itself, you can let the body move and make curved surfaces and so on with no problem. I was simply trying to indicate that people with little experience can start off very simply and still do some interesting problems.

H. Lugt

Your pictures are very impressive, but I am a bit concerned about the accuracy. Could you comment on that?

C. W. Hirt

Certainly. Accuracy, of course, depends upon the problem. You, for instance, are concerned with the accuracy of flow around the corners of very thin discs and plates. In our case, where we are concerned with large blunt bodies with sharp corners and separation at the corners, the accuracy when compared with experimental data is pretty good. So, I guess a proper answer to your question is that we can get good results in some cases, but certainly not in others.

T. Taylor

I realize that you have to sacrifice some accuracy in order to make a movie such as you have shown us. But, it would be nice if you compared pressures, velocities and so on in order to tell whether it is a qualitative calculation. It seems to me that this is the jist of the question.

C. W. Hirt

Yes, the collapsing wake problem is an example. The very first case was a benchmark. It was compared with Wu's experiments and agreed very well. Such results give you confidence to go on and look at the variations.

T. Taylor

Well, your movies are impressive and I think that anyone who has done these has reservations about the fine detail.

N. Salvesen

I would like to stick with the accuracy question a bit longer. You are at the point of deciding where to go from here and I wonder if that isn't a bit premature. You haven't demonstrated that we can really solve physical problems.

T. Taylor

I don't want the audience to get the impression that these calculations are not correct. They do give some insight into the physics even if in a qualitative fashion. I agree that you won't get pressure or velocity, but you do get some feel for the mechanics. And, the calculations do simulate inertia effects pretty nicely. Now, whether they get all the viscous effects just right or whether they produce an artificial effect is another question. But, qualitatively I think we might learn something from doing these calculations.

C. W. Hirt

This discussion of accuracy leads me to believe that I may have been misunderstood. I agree completely with what has just been said. What I am arguing for is application of a tool that has demonstrated its potential. Certainly it may need refining and adjustment. And, those who will apply the technique should provide those refinements and adjustments during the course of their calculations.

G. Birkhoff

This work, of course, is very interesting. However, I wonder if we need a flood of numerical data at this particular time. The most difficult question consists of knowing how to interpret such data. However, I do think that some carefully monitored codes and benchmark data comparing, say, model experiments would be very worthwhile.

J. Enig

Several years ago we looked at the problem of a right-circular cylinder impacting on water. We looked at the diameter of the outer-most lip of the splash and plotted that as a function of depth of penetration into the water. We found that with the limited penetration in our experiment, the diameter of the lip of the splash was in excellent agreement with the experiments done by May and others at the Naval Ordnance Laboratory. I don't know if it was fortuitous, but it came out perfect. I bring this out because people have questioned the accuracury of these marker-and-cell calculations.

P. Roache

On this question of accuracy, I think it is not just the technique but the problem you look at. From what I've been able to see, when you have a free surface problem dominated by inviscous effects of the surface your accuracy is very good. And, if you have a viscous problem in which the boundary layer approximations are very good, even though you may be using the Navier-Stokes equation, you would come out very well. But,

that may give you a false sense of security about your code and about the accuracy of the technique. On the other hand, if you get into some of the problems mentioned this morning, such as those involving viscous separation where all the viscous stress terms are important, then you will find that with high Reynolds number flow you are going to get very inaccurate results.

J. Boris

I would like to add a few comments about accuracy. The impression given is that you can just take these 2- and 3-D codes which give qualitative impressions of what the flow is like and apply them to hydrodynamic problems. I have spent quite a lot of time with difference schemes, (the same analyses probably also apply to finite element methods or modal treatments), and there is a limited resolution that you can get in an approximation to a problem. Look at the errors you get in finite difference treatments, phase errors, diffusion errors and errors due to having a finite resolution. Generally speaking, there seems to be no way that you can do better than a mode by mode elimination of the phase and diffusion errors, at least in a linear sense. Accuracy can be improved by increasing the number of modes. But, in the primitive case at least, the actual Reynolds number of the flow that can be represented, starting with the basic equation, is very limited. There is no reason to believe that flows with 20 or 30 cells will actually reproduce the 5 to 10 per cent accuracy that is being required from the flow. The inate error due to resolution means that you can do a perfect job on the amplitude and on the phase relationship of harmonic motion and you still have an appreciable error. That error is irreducible in the sense that you need more points, more modes or, if you want, more trial functions in order to reduce that error. Some methods are better than others, but in everything that I have tried you keep coming up against Reynolds numbers of a hundred, twenty, fifty, and trying to push that number very high is extremely difficult.

T. Taylor

Are you saying that if you try to push the number higher you are getting instability?

J. Boris

No, its just that the resolution limits the Reynolds number that you can model.

A question was just asked about the stretching of the coordinates, and that is fine with lagrangian systems. But you are still limited, in the final analysis, by the number of zones that can be represented in a computer. If you have very complicated flow then you have to get a very complicated coordinate for stretching, and I don't think that will help very much.

T. Taylor

Are you saying that spectral schemes encounter the same problem?

J. Boris

In the final analysis they do although they will be better because they can eliminate the dispersion error and the diffusion error in finite difference schemes. But the resolution problem remains. So, in order to do a good job in solving the equations they have to put in a large enough viscosity that the physical Reynolds number dominates the numerical Reynolds number in the problem; otherwise they are being dominated by the numerical effects on top of the physical.

PROGRESS IN COMPUTATIONAL FLUID DYNAMICS
AT NASA AMES RESEARCH CENTER

Robert W. MacCormack
NASA Ames Research Center
Moffett Field, California

INTRODUCTION

During the past decade a revolution has been taking place in the study of fluid dynamics. The computational branch of this discipline has greatly increased its power and has gained a large share of respectability from the fluid dynamics community. In the early 1960's computers were just beginning to be powerful enough for the practical simulation of two-dimensional flows. The numerical techniques in use then, appear now by our present standard to be much like dinosaurs -- consuming large amounts of computing time and producing few results. During the decade as major advances were being made in computer technology, rapid progress was also being made in the development of efficient and reliable numerical techniques. Today NASA and the aerospace industry are relying heavily on numerical fluid dynamics because it is both good business and good science.

It is good business because numerical flow simulations are becoming less expensive each year and many are now more economical than experimental simulation. Computer technology has increased computing speeds by a factor of about ten every three years. This has resulted in a reduction of the computation cost of a given problem (Figure 1) by a factor of ten approximately every five years. At the same time the cost of wind tunnel tests for aircraft development (Figure 2) has been dramatically increasing each year. Because of these trends, the computer will be playing an ever increasing role in assuming wind tunnel

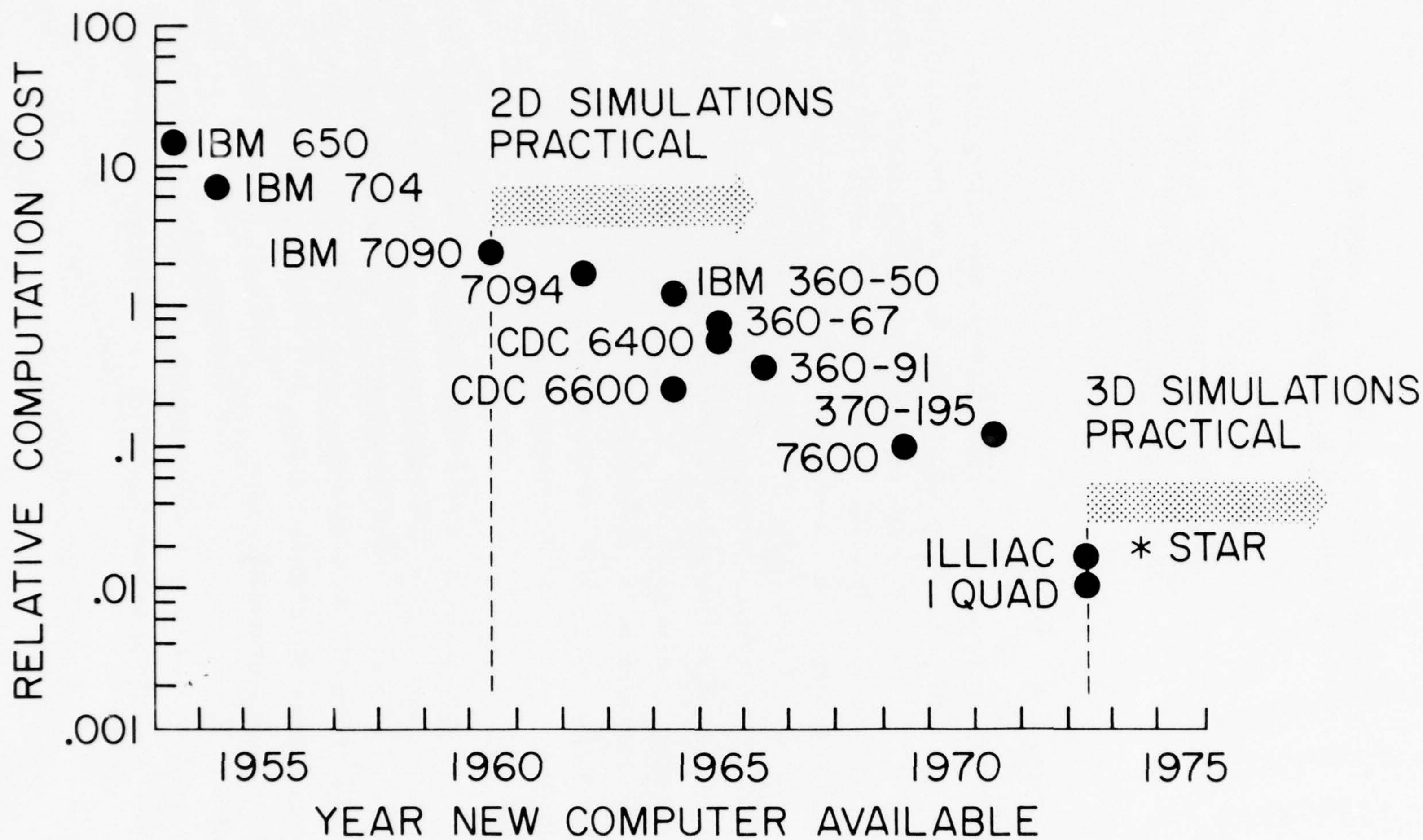


Figure 1. Trend of Computation Cost For Computer Simulation of a Given Flow.

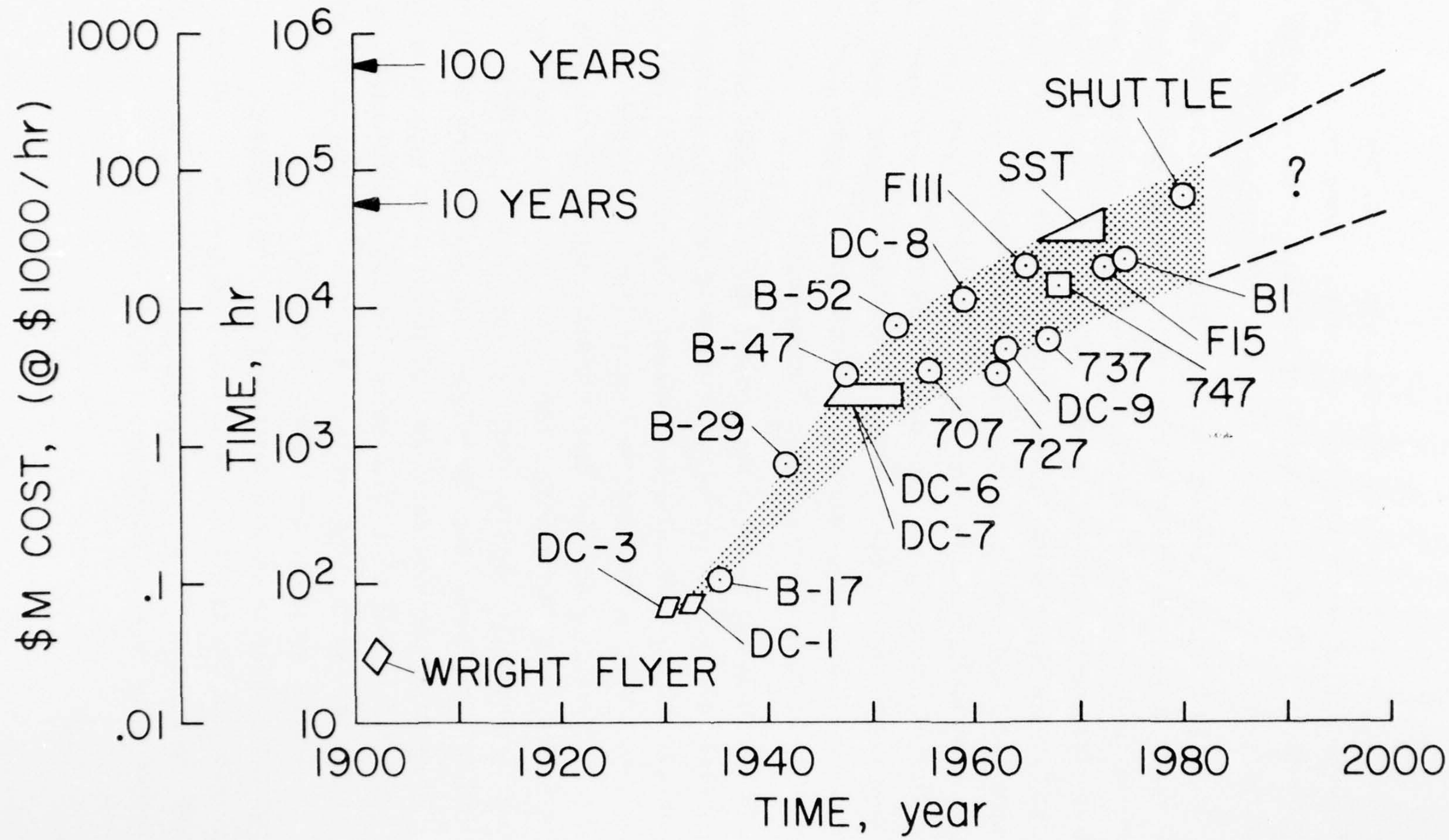


Figure 2. Total Wind Tunnel Test Hours for Development of Various Aircraft.

tasks. Already it is being used to reduce wind tunnel testing by the early elimination of unpromising designs found by numerical simulation. A recent example is in the development of variable camber wings for transonic speeds.

It is good science because numerical flow simulations can provide information not obtainable by other means. These include flows outside the testing range of experimental facilities, such as flows at flight Reynolds numbers and with flight air chemistry, and also flows unaffected by the experimental apparatus itself, such as those free of wind tunnel wall and model support interference effects. In addition to these, numerical simulation can provide insight for understanding the dynamics of complex flows. For example, at Ames in a combined computer wind tunnel investigation studying turbulence, several analytical turbulence models are being numerically simulated and compared with experiment to further understand this phenomenon.

In the following sections the recent progress at Ames in computational fluid dynamics for three major areas of research; supersonic, transonic and viscous flow, will be reviewed. The numerical techniques for these areas will be described in general terms only. References will be given providing detailed descriptions. Not all of these techniques were devised or developed at Ames. Many have come from other government laboratories, private industry, and universities both in the United States and abroad. Some techniques have a wide range of application. For example, one originally devised to calculate hypervelocity impact cratering in solids[1] is also being used today to calculate the reentry flow field about the space shuttle orbiter[2] as well as the flow within the exhaust system of the Yamaha motorcycle engine.[3] Others have application only to solve a specific type of gas dynamics problem. Finally, although the techniques to be described and their applications represent the state of the art today, they will probably and even hopefully appear to be much like dinosaurs a decade from now.

SUPERSONIC FLOW

The numerical simulation of flows past bodies traveling at supersonic speeds has progressed very rapidly. Only a few years ago we were just beginning to calculate supersonic perfect gas flows past bodies of simple conical geometries.[4] Today we can simulate the three-dimensional steady inviscid supersonic flow, for either a perfect or real gas, past a complete aircraft-like configuration.[5] These complex flows, in general, contain (Figure 3) a detached bow shock wave, embedded canopy and wing shocks, expansions and recompression shocks on the lee side of the body, and contact discontinuities at shock wave intersections. The governing steady Euler equations for such flows are elliptic in the region between the bow shock wave and the body at the nose where the flow is subsonic and are hyperbolic in the stream direction elsewhere. These two distinct types of flow require separate numerical treatment. The elliptic flow near the nose is calculated using techniques for blunt body flows. After the nose region solution is calculated, it is used as an initial condition for the remaining purely supersonic flow.

Blunt Body Flow

A supersonic flow past a blunt body contains a detached shock wave which separates the disturbed flow from the free stream. The shock wave is at some point normal to the free-stream direction and degenerates into a Mach wave away from this point. On the body surface there is a stagnation point away from which the flow expands until again reaching supersonic speeds. The flow contains an embedded subsonic region lying between the shock wave and the body surface and also between the two sonic lines each extending from the body to the shock across which the flow becomes supersonic. Mathematically the flow is described by an elliptic set of equations in the embedded subsonic region and by a hyperbolic set elsewhere. Computationally the flow is difficult to calculate because of the range of Mach numbers encountered, from zero to in excess of one, the shock wave discontinuity, and the body geometry.

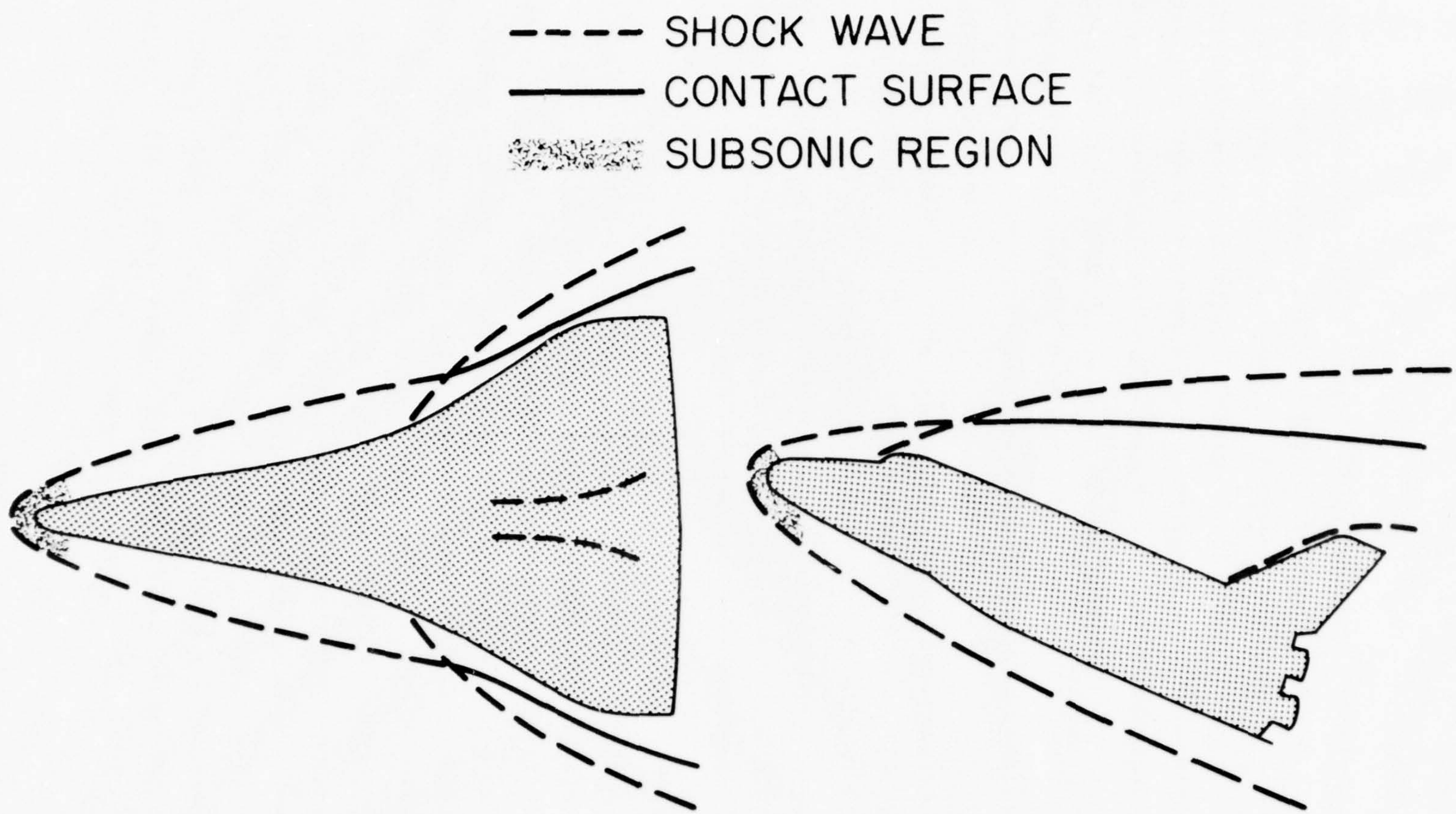


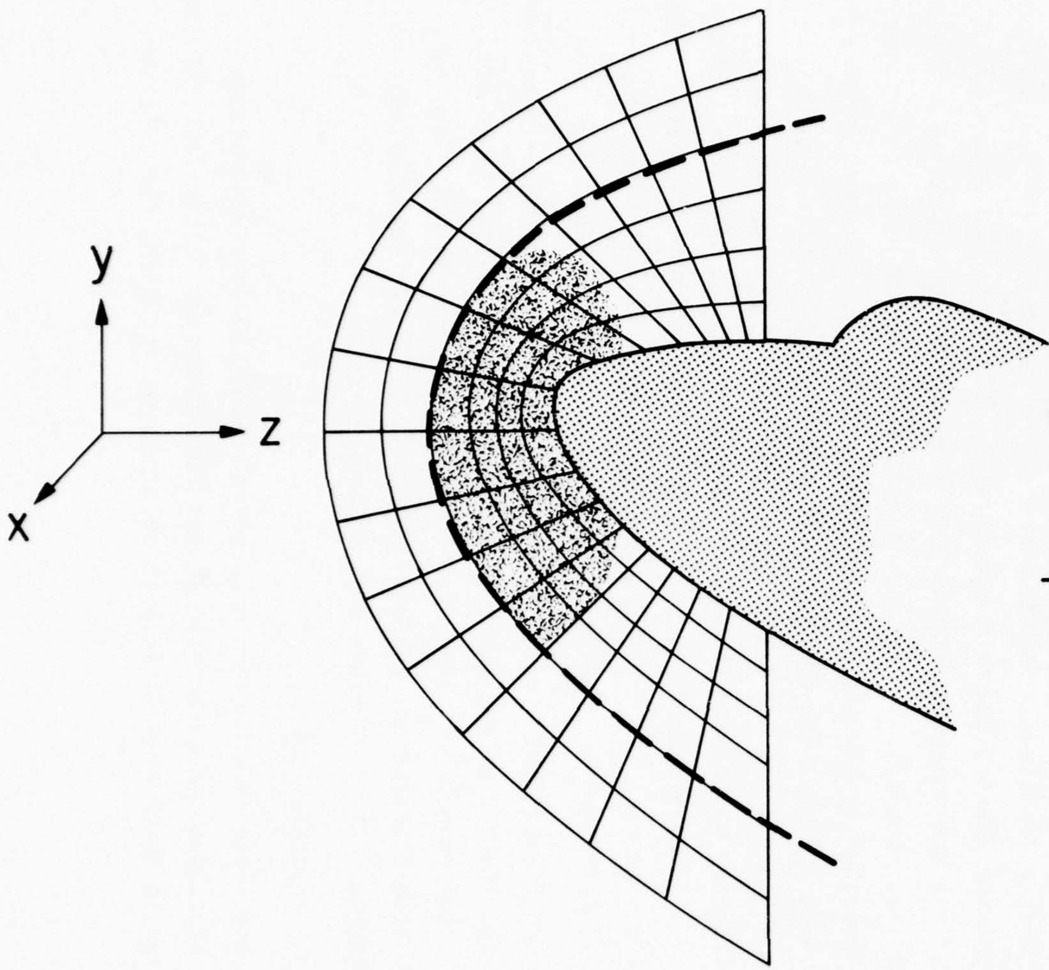
Figure 3. Three-Dimensional Supersonic Flow.

Although in general only the steady state solution is desired, the unsteady Euler equations in integral form are chosen for numerical solution. The time dependent equations are everywhere hyperbolic in time and no distinction between subsonic and supersonic regions need be made. Solutions to the steady flow equations are approached asymptotically in time. The integral form of the equations enables both the body surface and shock wave discontinuity to be treated in a straightforward manner. The volume between the shock wave and body is discretized into a set of arbitrarily shaped small volumes (Figure 4). This set constitutes the finite difference mesh upon which the integral equations are numerically solved. As the shock wave moves during the calculation the unsteady Rankine-Hugoniot shock relations are used to adjust the mesh to fit the changing shape of the shock. The conservative variables of density, momentum, and energy are advanced in time using numerical approximations to the net flux and stress at each surface enclosing a small mesh volume. The equations are approximated to second-order accuracy in both space and time. Rizzi and Inouye[6] of Ames using this approach have calculated three-dimensional flows past bodies with space shuttle-like nose geometries. In Figure 5 their theoretical results for a sphere-cone body (23° half cone angle) in Mach 14.9 flow are compared with the experimental results of Cleary and Duller of Ames.

Other excellent blunt-body programs using transformed differential equations instead of the integral form have been developed by Moretti and his coworkers[7,8] at the Polytechnic Institute of Brooklyn and by Li[9] of Lockheed Electronics, Company, Houston. Li is presently extending his program[10] to treat blunt body viscous flows.

Purely Supersonic Flow

After the flow becomes supersonic again, the governing steady Euler equations are hyperbolic in the flow direction. Therefore, if the solution is known at a surface at one upstream location, the solution



CONTINUITY EQUATION
(UNSTEADY)

$$\frac{\partial \rho}{\partial t} = - \frac{\partial \rho u}{\partial x} - \frac{\partial \rho v}{\partial y} - \frac{\partial \rho w}{\partial z}$$

Figure 4. Blunt Body Region.

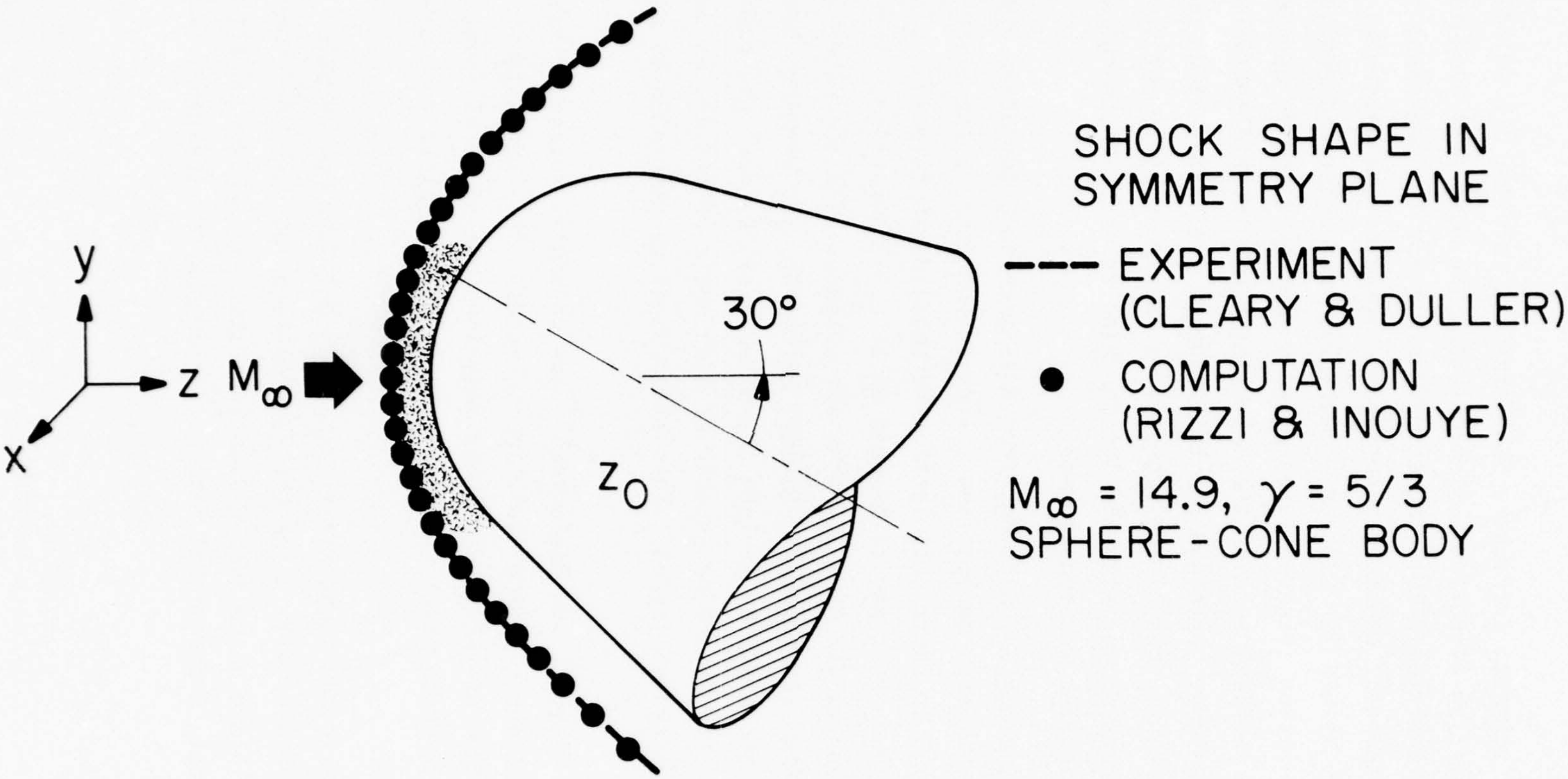


Figure 5. Comparison of Computational and Experimental Results for Blunt Body Flow Region.

can then be determined everywhere downstream by marching the solution surface in the flow direction. At each step, the numerical technique approximating the steady Euler equations solves for the dependent flow variables at the next downstream surface in terms of the variables at the previously solved for surface (Figure 6). For a three-dimensional flow the calculation is actually only two-dimensional and is updated in space. In the last section the calculation was three-dimensional and was updated in time. Although the technique of the last section could be used to solve for the entire flow, it is much more efficient for steady flow problems to switch to the marching in space technique as soon as the flow becomes supersonic.

Kulter, Lomax and Warming[2] from Ames developed the above approach to solve for the very complex flow fields encountered by the space shuttle on reentry. In their calculation the solution surface is a plane normal to the body axis and is marched in an axial direction. Their theoretical results for Mach 7.4 flow are compared with the experiment of Cleary at Ames in Figure 7. Kutler, Reinhardt, and Warming[5] have extended this technique to include the effects of the real gas chemistry encountered by the returning shuttle at high angle of attack, extreme altitude and hypersonic speed. This flow cannot be simulated by ground experiment.

Rizzi, Klavins, and MacCormack[11] are presently extending Kutler's approach to include nonplanar solution surfaces. This extension using the integral form of the steady Euler equations can treat all supersonic flows, not just those which are always supersonic relative to a given fixed direction.

In both Kutler's and Rizzi's approach only the bow or outer shock wave is treated specially using the Rankine-Hugoniot jump relations to accurately maintain sharp jump discontinuities. In another excellent approach Moretti, Grossman and Marconi[12] give special treatment to all

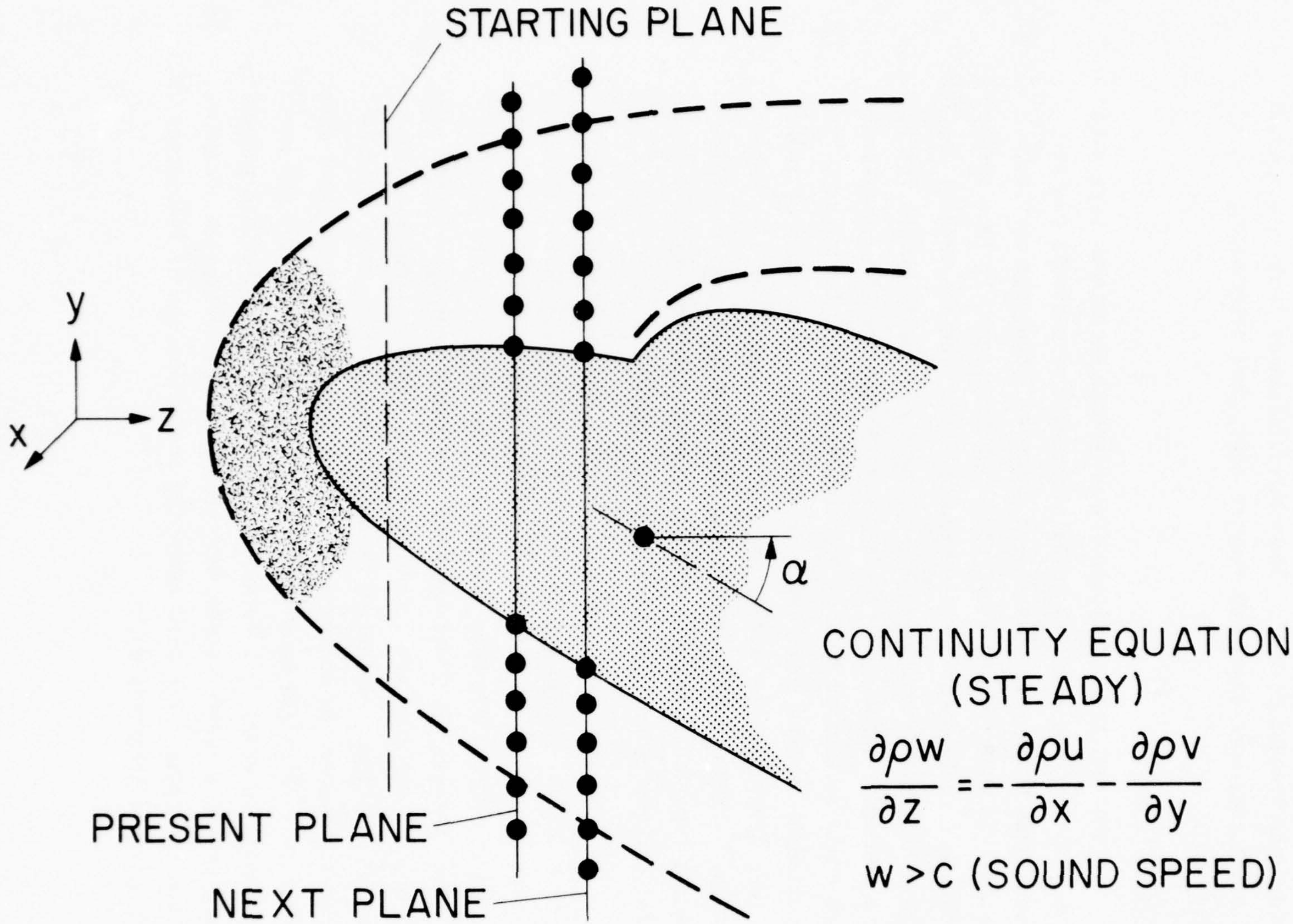


Figure 6. Supersonic Region

embedded shock waves as well. Their calculation of these shock waves are, of course, more exact but require additional programming logic beforehand.

TRANSONIC FLOW

In the last section hyperbolic numerical techniques were used for supersonic flows possibly containing embedded subsonic regions. The equations which were solved were themselves hyperbolic; i.e., hyperbolic in time for the blunt body flow or hyperbolic in a spacial coordinate direction for pure supersonic flow. For the present section the governing steady equations are everywhere elliptic except for perhaps an embedded supersonic region. Even though the blunt body techniques could be applied to this problem, a different technique called relaxation has proven to be far more efficient. Techniques of this type were originally devised to solve elliptic equations like Laplace's equation.[13] The equations describing nearly sonic flows, resemble Laplace's equation and can be relaxed in much the same way if certain special procedures are used in supersonic regions.

The progress in this area of research has proceeded amazingly fast since 1970. Murman and Cole[14] at the Boeing Scientific Research Laboratories and Steger and Lomax[15] of Ames made important early contributions. Murman and Cole devised a clever change in the elliptic difference operator to treat embedded supersonic regions. Steger and Lomax quickly extended the technique to treat the full compressible potential equations. Recently, Murman[16] since joining Ames has made another improvement by modifying the difference equation at shock waves to accurately calculate shock position and strength. Martin and Lomax[17] are presently developing a technique using fast direct Poisson solvers which promises a further speed improvement of the relaxation procedure. Ballhaus and Lomax[18] are also extending the advantages of relaxation to treat unsteady transonic flows.

Bailey and Ballhaus[19,20] have developed methods to calculate the three-dimensional flow fields about wings and wing-body combinations. They solve the small disturbance equations and can treat wings with sweep, taper, twist and camber. The results of their calculations for a C-141 wing planform are shown in Figures 8 and 9. In Figure 8 is shown the computed isobars on the top surface of the wing. In Figure 9 is a comparison of the computed results with both flight and wind tunnel measurements. The tunnel, which had Reynolds number limitations, did not simulate the flight conditions as well as the inviscid numerical calculation.

Another excellent technique for three-dimensional transonic flows developed by Jameson[21] of the Courant Institute of New York University uses the full potential equations. It is presently being used at Ames to calculate the flow past yawed wings.

VISCOUS FLOW

As in the inviscid supersonic and transonic cases, the numerical simulation of viscous flows has also progressed rapidly. The simulation of flows governed by the compressible form of the Navier-Stokes equations is a good example. Just a few years ago we were able to begin solving for two-dimensional shock-induced separated laminar boundary layer flows at Reynolds numbers of 10^5,[22,23] and today we are on the threshold of extending our calculations to treat turbulent boundary layers at Reynolds numbers of 10^7. These calculations use predominantly if not solely explicit finite difference schemes for solving the governing mixed hyperbolic and parabolic differential equations. The calculations for flows with high Reynolds numbers require large amounts of computer time, even with the latest generation of computers, basically because explicit schemes are subject to the Courant-Friedrichs-Lewy stability condition on the maximum time step size. This condition limits the time step to be proportional to the mesh point spacing which is in viscous shear

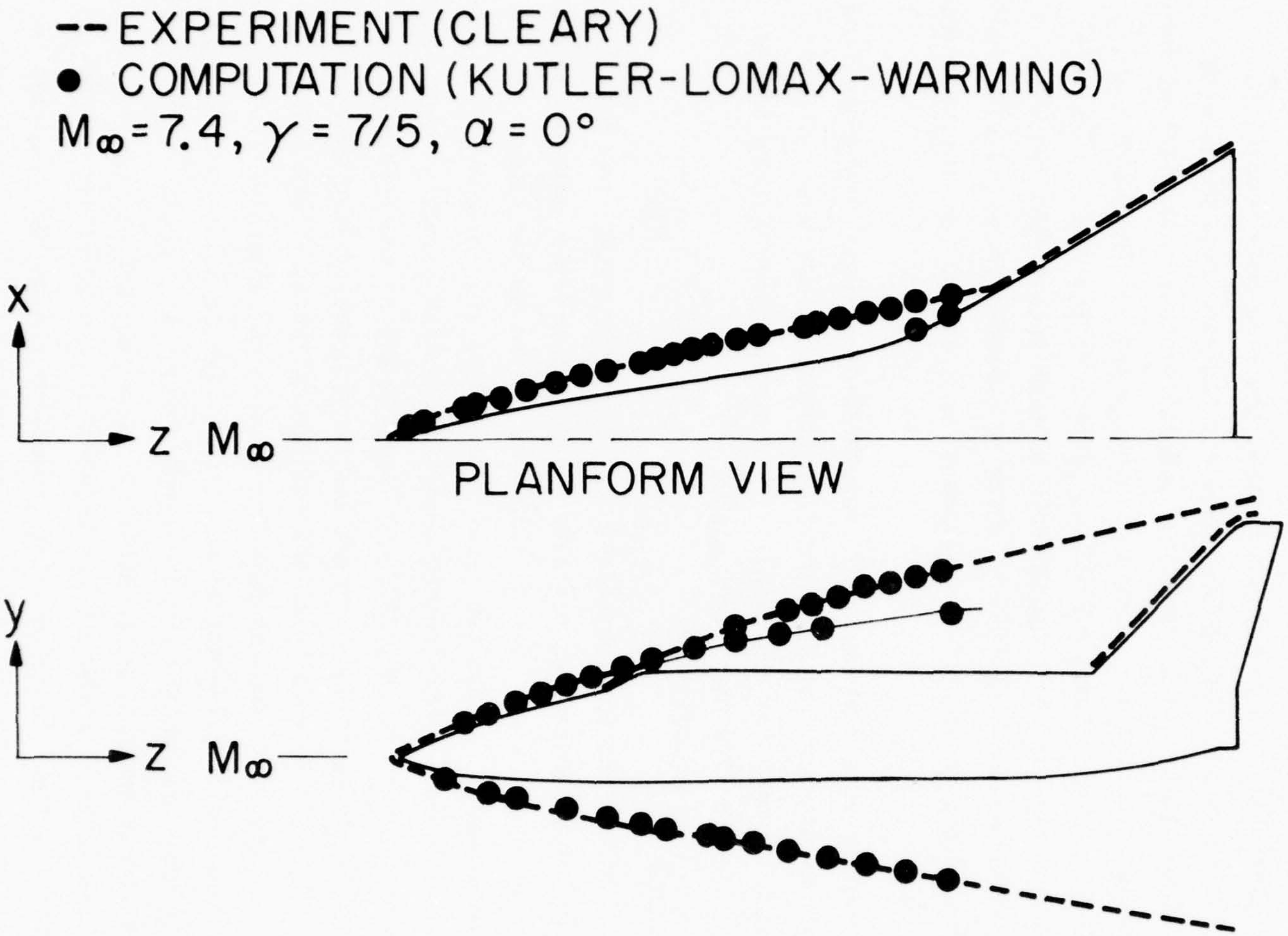


Figure 7. Comparison of Computational and Experimental Results for Supersonic Flow Region.

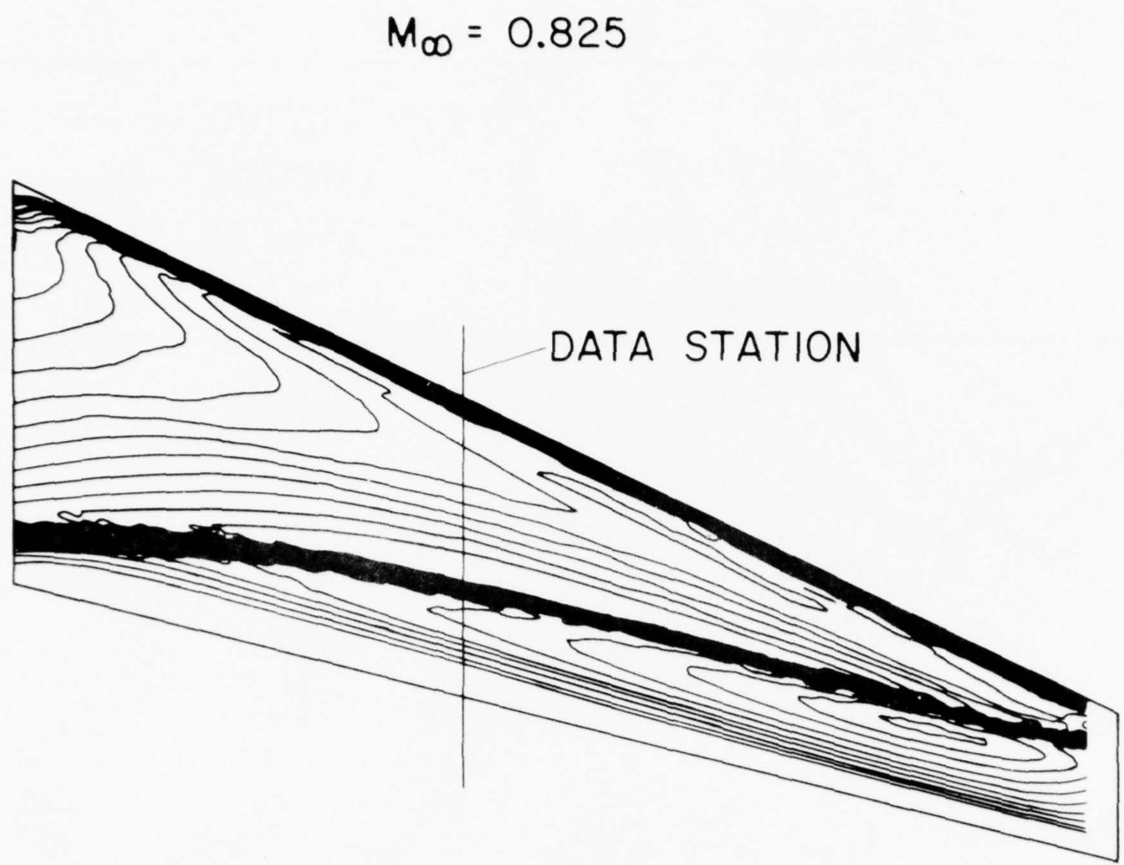


Figure 8. Computed Isobars on C-141 Platform at Mach 0.825.

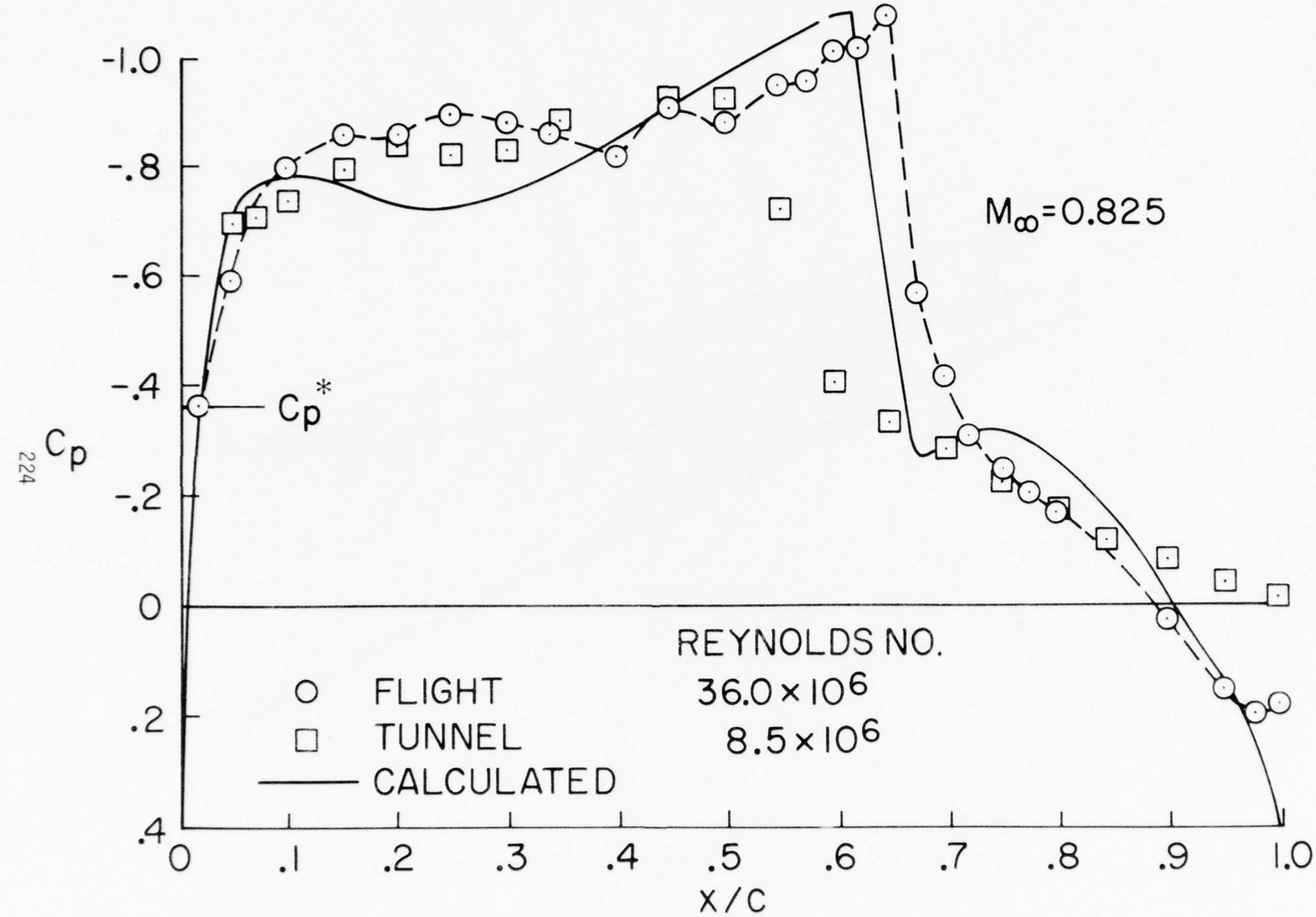


Figure 9. Wind Tunnel, Flight, and Computed Data for Upper Surface of C-141 Wing at Mach 0.825.

regions of the order of the reciprocal of the square root of the Reynolds number. Recently several implicit schemes,[24,25] which are free from the restrictive CFL condition, have appeared in the literature for solving mixed hyperbolic and parabolic systems. Though none of these has yet been tested by the fire of calculating shock-boundary layer interactions at high Reynolds numbers, they have been able to operate stably and satisfactorily with time step sizes orders of magnitude larger than that allowed for explicit schemes. These schemes show promise of a large increase in computational efficiency and may enable us to extend our calculations to three-dimensional high Reynolds number flows with our present computers.

For the incompressible Navier-Stokes equations there are several good techniques. These include the technique developed by Chorin[26] while at the Courant Institute which solves these equations in the primitive variables of velocity and pressure and the technique of Thomson and Shanks of Mississippi State University and Wu[27] of Georgia Institute of Technology which solves the vorticity form of the equations. In addition to these Chorin[28] has recently developed at the University of California, Berkeley, a new technique which does not use a finite difference mesh. The technique calculates the vorticity required at the body surface to satisfy the no-slip condition and then allows this vorticity to be shed into the flow as discrete point vortices. The point vortices then interact with each other as they convect away from the body and determine the entire fluid velocity field.

Presently at Ames we are applying both the recent Chorin technique and an implicit finite difference technique developed by Mehta and Lavan[29] at the Illinois Institute of Technology to calculate incompressible two-dimensional flows containing large regions of separation, for example, airfoil stall.

Baldwin and MacCormack[30,31] have calculated the interaction of a strong shock wave with a hypersonic turbulent boundary layer at high Reynolds numbers. These calculations require the use of models for turbulence closure because of the broad range of time and length scales existing in turbulent eddying flows. Finite difference calculations are limited in both spacial and temporal resolution to the sizes of their mesh spacing and time step increments and, hence, models are required to account for the sub-grid size eddies and high frequency fluctuations. Much research is now going on in both the United States and abroad to model this phenomenon. Baldwin and MacCormack are presently testing both simple mixing length and the more complex two equation transport models. Deiwert[32] in a related approach at Ames is calculating shock-induced and trailing-edge separation of turbulent boundary layers for transonic flows past thick airfoils at Reynolds numbers as high as ten million. His approach uses the integral form of the Navier-Stokes equations which can conveniently treat general airfoil shapes.

CONCLUSIONS

During the last decade we have witnessed a considerable amount of progress in computational fluid dynamics. This progress has enabled us to extend our two-dimensional inviscid supersonic and transonic flow calculations to three-dimensional flows past wing-body combinations using about the same computer time as before. In a sense our status for viscous flows is where we were for inviscid flows a decade ago. It is not unexpected that a decade from now it will be practical to numerically simulate high Reynolds number viscous flows about complete aircraft configurations.

REFERENCES

Supersonic Flow

1. MacCormack, R.W., "The Effect of Viscosity in Hypervelocity Impact Cratering," AIAA Paper Number 69-354, 1969.

2. Kutler, P., H. Lomax and R.F. Warming, "Computation of Space Shuttle Flow Fields Using Noncentered Finite-Difference Schemes," AIAA Journal, Volume 11, Number 2, pp. 196-204, February 1973.

3. Dwyer, H., R. Allen, M. Ward, D. Karnopp and D. Margolis, "The Use of Shock Capturing Finite Difference Methods for Unsteady Gas Transfer Problems," AIAA Paper 74-521, 1974.

4. Kutler, P. and H. Lomax, "Shock-Capturing, Finite-Difference Approach to Supersonic Flows," Journal Spacecraft Rockets, Volume 8, Number 12, pp. 1175-1182, December 1971.

5. Kutler, P., W.A. Reinhardt and R.F. Warming, "Multishocked, Three-Dimensional Supersonic Flowfields with Real Gas Effects," AIAA Journal, Volume 11, Number 5, pp. 657-664, May 1973.

6. Rizzi, A.W. and M. Inouye, "A Time Split Finite Volume Technique for Three-Dimensional Blunt Body Flow," AIAA Journal, Volume 11, November 1972.

7. Moretti, G. and M. Abbett, "A Time-Dependent Computational Method for Blunt Body Flows," AIAA Journal, Volume 4, pp. 2136-2141, 1966.

8. Moretti, G. and G. Bleich, "Three-Dimensional Flow Around Blunt Bodies," AIAA Journal, Volume 5, pp. 1557-1562, 1967.

9. Li, C.P., "Time-Dependent Solutions of Nonequilibrium Airflow Past a Blunt Body," AIAA Paper 71-595, 1971.

10. Li, C.P., "A Numerical Study of Laminar Flow Separation on Blunt Flared Cones at Angles of Attack," AIAA Paper 74-585, 1974.

11. Rizzi, A.W., A. Klavins and R.W. MacCormack, "A Generalized Hyperbolic Marching Technique for Three-Dimensional Supersonic Flow with Shocks. Paper to be presented at the 4th International Conference on Numerical Methods in Fluid Dynamics, Boulder, Colorado, June 24-29, 1974.

12. Moretti, G., B. Grossman and F. Marconi, Jr., "A Complete Numerical Technique for the Calculation of Three-Dimensional Inviscid Supersonic Flows," AIAA Paper 72-192, 1972.

Transonic Flow

13. Isaacson, E. and H.B. Keller, "Analysis of Numerical Methods," John Wiley & Sons, 1966.

14. Murman, E.M., and J.D. Cole, "Calculation of Plane Steady Transonic Flows," AIAA Journal, Volume 9, Number 1, pp. 114-121, 1971.

15. Steger, J.L. and H. Lomax, "Transonic Flow About Two-Dimensional Airfoils by Relaxation Procedures," AIAA Journal, Volume 10, Number 1, pp. 49-54, 1972.

16. Murman, E.M., "Analysis of Embedded Shock Waves Calculated by Relaxation Methods," Proceedings of AIAA Computational Fluid Dynamics Converence, Palm Springs, California, July 1973, pp. 27-40.

17. Martin, E.D. and H. Lomax, "Rapid Finite-Difference Computation of Subsonic and Transonic Aerodynamic Flows," AIAA Paper 74-11, 1974.

18. Ballhaus, W.F. and H. Lomax, "The Numerical Simulation of Low Frequency Unsteady Transonic Flow Fields," Paper to be presented at the 4th International Conference on Numerical Methods in Fluid Dynamics, Boulder, Colorado, June 24-29, 1974.

19. Bailey, F.R. and W.F. Ballhaus, "Relaxation Methods for Transonic Flow About Wing-Cylinder Combinations and Lifting Swept Wings," Lecture Notes in Physics, Volume 19, Springer-Verlag, pp. 2-9, 1972.

20. Ballhaus, W.F. and F.R. Bailey, "Numerical Calculation of Transonic Flow About Swept Wings," AIAA Paper 72-677, June 1972.

21. Jameson, A., "Numerical Calculation of the Three-Dimensional Transonic Flow Over a Yawed Wing," Proceedings AIAA Computational Fluid Dynamics Conference, July 1973, pp. 18-26.

Viscous Flow

22. MacCormack, R.W., "Numerical Solution of the Interaction of a Shock Wave with a Laminar Boundary Layer," Lecture Notes in Physics, Volume 8, Springer-Verlag, New York, 1971, p. 151.

23. Carter, J.E., "Numerical Solutions of the Supersonic Laminar Flow Over a Two-Dimensional Compression Corner," Lecture Notes in Physics, Volume 19, Springer-Verlag, New York, 1973, p. 69.

24. Baum, E. and E. Ndefo, "A Temporal ADI Computational Technique," Proceedings AIAA Computational Fluid Dynamics Conference, Palm Springs, California, July 1973, pp. 133-140.

25. Briley, W.R. and H. McDonald, "An Implicit Numerical Method for the Multidimensional Compressible Navier-Stokes Equations," United Aircraft Corporation, Report M911363-6, November 1973.

26. Chorin, A.J., "Numerical Solution of the Navier-Stokes Equations," Math. Comp., Volume 22, Number 104, 1968, pp. 745-762.

27. Thompson, J.F., S.P. Shanks and J.C. Wu, "Numerical Solution of Three-Dimensional Navier-Stokes Equations in Integro-Differential Form: Flow About a Body," Proceedings AIAA Computational Fluid Dynamics Conference, Palm Springs, California, July 1973, pp. 123-132.

28. Chorin, A.J., "Numerical Study of Slightly Viscous Flow," Journal Fluid Mech., Volume 57, Part 4, 1973, pp. 785-796.

29. Mehta, U.B. and F. Lavan, "Starting Vortex, Separation Bubbles, and Stall -- A Numerical Study of Laminar Unsteady Flow Around an Airfoil," Paper submitted to Journal Fluid Mech., 1974.

30. Baldwin, B.S. and R.W. MacCormack, "Numerical Solution of Interaction of Strong Shock Wave with a Turbulent Boundary Layer," AIAA Paper 74-558, 1974.

31. Baldwin, B.S. and R.W. MacCormack, "Interactions of Strong Shock Wave with Turbulent Boundary Layer," Paper to be presented at the 4th International Conference on Numerical Methods in Fluid Dynamics, Boulder, Colorado, June 24-29, 1974.

32. Deiwert, G.S., "Numerical Simulation of High Reynolds Number Transonic Flows," Paper to be presented at the 4th International Conference on Numerical Methods in Fluid Dynamics, Boulder, Colorado, June 24-29, 1974.

NUMERICAL SIMULATION OF TURBULENT FLOWS

Steven A. Orszag
Department of Mathematics
Massachusetts Institute of Technology
Cambridge, Massachusetts

ABSTRACT

This paper reviews work on direct numerical simulation of turbulent flows by solution of the Navier-Stokes equations. Limitations on spatial and temporal resolution and boundary and initial conditions are considered. The current state of the art of simulations of homogeneous and shear turbulence is discussed, as well as the prospects for future simulations based on realistic extrapolations of available computer resources.

INTRODUCTION

Digital computers capable of 1-10 MIPS (millions of instructions per second) are now readily available (e.g., CDC 6600, 7600, IBM 360-195), while machines currently being developed are hoped to be capable of 10-100 MIPS (CDC STAR, Illiac, TI ASC, Cray Research). The great power of these machines has made possible the solution of some of the most challenging fluid dynamical problems, those of turbulence, by numerical solution of the Navier-Stokes equations.

Early progress towards this goal was reported by Orszag and Patterson (1972).[22] In this work, simulations of three-dimensional homogeneous isotropic turbulence at moderate Reynolds numbers were performed and the results compared with the predictions of turbulence theory.

Since then several studies have been made of other turbulent flows using similar methods. In the present paper, we review work on these calculations.

In the next section, we summarize the numerical methods we have employed in our computations. Under PROBLEMS, some of the current limitations of our methods and fundamental difficulties encountered to date are discussed in order to lend perspective to the range of possible applications. Under SURVEY OF APPLICATIONS, a survey of some applications is given. Then, under COMPARISON WITH OTHER METHODS, the direct simulation approach discussed here is contrasted with modelling turbulence by statistical approximation (Launder and Spalding 1972,[13] Harlow 1973,[5] among others). Finally, under PROSPECTS, we discuss the effect of recent computer developments and try to forecast reasonable expectations of the simulation approach over the next few years.

METHODS

The principal problems of interest involve incompressible flows governed by the Navier-Stokes equations

$$\frac{\partial \vec{v}(\vec{x},t)}{\partial t} + \vec{v}(\vec{x},t)\cdot\vec{\nabla}\vec{v}(\vec{x},t) = -\vec{\nabla}p(\vec{x},t) + \nu\nabla^2\vec{v}(x,t) \qquad (1)$$

$$\vec{\nabla}\cdot\vec{v}(\vec{x},t) = 0 \qquad (2)$$

where $\vec{v}(\vec{x},t)$ is the velocity field, $p(\vec{x},t)$ is the pressure, and ν is the kinematic viscosity. Equations (1) and (2) isolate the basic non-linear mechanisms of turbulent flow; other effects may be important including compressibility, buoyancy, chemical reactions, multi-phase flows, etc. It is appropriate to recall that it follows from (1) and (2) that

$$\nabla^2 p(\vec{x},t) = -\vec{\nabla}\cdot[\vec{v}(\vec{x},t)\cdot\vec{\nabla}\ \vec{v}(\vec{x},t)]; \qquad (3)$$

this Poisson equation determines the pressure in order that the incompressibility constraint (2) be satisfied. The pressure is governed by the diagnostic equation (3) rather than a prognostic equation for $\partial p/\partial t$, as would be the case in a compressible flow with finite sound speed. [In

fact, the pressure in (1) is quite analogous to a Lagrange multiplier for maintenance of the kinematical constraint (2).]

Homogeneous turbulence can be simulated by imposition of periodic boundary conditions in (1), i.e.

$$\vec{v}(\vec{x} + 2\pi\vec{n},t) = \vec{v}(\vec{x},t) \tag{4}$$

where $\vec{n}$ is a vector with integer components. With these boundary conditions, an attractive method for numerical solution of the Navier-Stokes equations is to seek an approximate solution of the form of a truncated Fourier series

$$\vec{v}(\vec{x},t) = \sum_{|\vec{k}|<K} \vec{u}(\vec{k},t)e^{i\vec{k}.\vec{x}} \tag{5}$$

where the wavevectors $\vec{k}$ must have integer components if the periodicity interval is 2π as in (4). When (5) is used, the Navier-Stokes equations become

$$\left(\frac{\partial}{\partial t} + \nu k^2\right)u_\alpha(\vec{k},t) = -ik_\beta\left(\delta_{\alpha\gamma} - \frac{k_\alpha k_\gamma}{k^2}\right) \sum_{\substack{|\vec{p}|<K \\ |\vec{k}-\vec{p}|<K}} u_\beta(\vec{p},t)u_\gamma(\vec{k}-\vec{p},t) \tag{6}$$

where summation over repeated Greek indices is implied and $\delta_{\alpha\gamma}$ is the Kronecker delta; the pressure has been eliminated from (6) by use of (3). Efficient techniques to solve the coupled system of ordinary differential equations in t are discussed by Orszag (1971).[15] This method of solving the Navier-Stokes equations is called a spectral method, in contrast to conventional finite-difference methods in which (1) and (2) are discretized on a finite grid of points.

More generally, spectral methods involve the representation of the velocity field by a truncated series in terms of smooth functions:

$$\vec{v}(\vec{x},t) = \sum_{|m|<M} \sum_{|n|<N} \sum_{|p|<P} \vec{a}_{mnp}(t)\psi_m(x_1)\phi_n(x_2)\chi_p(x_3) \tag{7}$$

The proper choice of expansion functions is crucial to the success of the method: the criteria are that there be rapid convergence to the exact solution as the cutoffs M, N,P $\to \infty$, and that there be efficient methods to solve the coupled system of differential equations for $\vec{a}_{mnp}(t)$ (Orszag and Israeli 1974).[19] Some appropriate choices of expansion functions are given in Table 1.

When these spectral methods are properly designed and implemented, they offer a number of advantages over finite difference methods. Five general areas of comparison are (Orszag and Israeli 1974):[19]

1. Rate of convergence - If $\vec{v}(\vec{x},t)$ is smooth (infinitely differentiable) then the error incurred by use of the spectral representation (7) goes to zero faster than any finite power of the cutoffs. In contrast, finite difference methods yield finite order rates of convergence (most usually second-order in the resolution). The important consequence is that high accuracy is achieved with little or no extra effort in spectral methods once moderate accuracy is achieved.

2. Efficiency - The development of fast transform methods has allowed spectral codes to be developed that are competitively fast as finite difference codes with the same number of independent degrees of freedom. However, the spectral codes require a factor 2-5 less degrees of freedom in each of the (three) space directions to resolve the flow with order 5% error. Hence, an order of magnitude or better improvement in utilization of computer resources is achieved.

3. Boundary conditions - There are three aspects to boundary conditions of importance for spectral methods. First, if Chebyshev or Legendre polynomials are used to represent the direction normal to a boundary layer of normalized thickness ε, only about $1/\sqrt{\varepsilon}$ polynomials need be retained to achieve high accuracy. On the other hand, a uniform grid with order $1/\varepsilon$ grid points would be required. Second, if coordinate transformations are employed to assist in the resolution of a boundary or internal layer of thickness ε, the spectral errors are decreased faster than any finite power

TABLE 1. CHOICE OF EXPANSION FUNCTIONS

	Type of Boundary Condition	Function	Comment
1.	Periodic	exp (ikx)	
2.	Free-slip (stress-free)	$\begin{Bmatrix}\sin\\ \cos\end{Bmatrix} kx$	Same as 1 with symmetry constraint
3.	Rigid (no-slip) a. Cartesian coordinates	$T_n(x)$	Chebyshev polynomials - series similar to cosine series since $T_n(\cos x) = \cos nx$
		$P_n(x)$	Legendre polynomials - less efficient than $T_n(x)$ but more natural for maintenance of conservation laws because orthogonality weight factor is 1 instead of $1/\sqrt{1-x^2}$
	b. Cylincrical (polar coordinates)	$r^s T_n(r)$	see Orszag (1974a)[16]
4.	Spherical polars	$Y_n^m(\theta,\phi)$	Surface harmonics -
		$\sin^s\theta \cos n\theta \exp(im\phi)$	Generalized Fourier series - see Orszag (1974a)[16]
5.	Semi-infinite and infinite geometry	$T_n(x)$	With mapping or truncation to finite domain
		$L_n(x)$	Laguerre polynomial $0 \leq x < \infty$
		$H_n(x)$	Hermite polynomial $-\infty < x < \infty$ see Orszag et al (1975)[18]

power of ε as $\varepsilon \to 0$. On the other hand, finite-difference results are improved by a finite power of ε. Third, with spectral methods high-order accuracy is achieved at inflow and outflow boundaries without the need for special methods to impose the boundary conditions. On the other hand, high-order difference methods break down near boundaries because grid points outside the physical domain must be invoked, and the necessary modifications to maintain accuracy near the boundary can get quite complicated (Kreiss and Oliger 1973).[12]

4. Discontinuities - Surprisingly, spectral methods do a better job of localizing errors than difference schemes and hence require considerably less local dissipation to smooth discontinuities.

5. 'Bootstrap' estimation of accuracy - It has been shown by Herring et al (1974)[7] that spectrum shape provides a built-in measure of accuracy of spectral calculations. If the spectral amplitudes approach zero regularly, it is possible to estimate the accuracy of the calculation from the results of the calculation itself. On the other hand, no such <u>internal</u> measure of accuracy of difference methods has yet been found; to determine the accuracy of a finite difference calculation, it is necessary to compare calculations of varying resolution.

Turbulence computations are performed by numerical solution of (1) and (2) together with suitable initial and boundary conditions. With homogeneous turbulence, periodic boundary conditions (4) are applied while random initial conditions are set up as follows. The spectral representation (5) is used with the choice

$$u_\alpha(\vec{k},0) = \left(\delta_{\alpha\gamma} - \frac{k_\alpha k_\gamma}{k^2}\right) r_\gamma(\vec{k}) \tag{8}$$

where $\vec{r}(\vec{k})$ are independent, Gaussianly distributed random variables with variance proportional to a specified (nonrandom) energy spectrum $E(\vec{k})$. The choice (8) guarantees the incompressibility constraint $k_\alpha u_\alpha(\vec{k},0) = 0$. When (8) is used in (5), there results a random initial flow field with Gaussian distribution and energy spectrum $E(\vec{k})$.

The establishment of random initial conditions for inhomogeneous turbulent flows like wakes and jets, is somewhat more complicated. It has been found (Orszag and Pao 1974)[21] that the most convenient way to maintain the incompressibility constraint is to express the velocity in terms of a vector potential

$$\vec{v}(\vec{x},0) = \vec{\nabla}\times\vec{A}(\vec{x}) \tag{9}$$

where $\vec{A}(\vec{x})$ is chosen to model the inhomogeneous statistics.

PROBLEMS

The fundamental difficulty with numerical simulation of turbulence can be illustrated by considering the possibility for calculating a flow similar to that observed by Grant, Stewart and Moilliet (1962)[4] in their study of the inertial-range spectrum of the flow in Discovery Passage, a tidal channel in British Columbia. The parameters of this flow are roughly:

$\overline{v}$ = RMS turbulent velocity	$\sim$	150 cm/s
L = Large eddy size	$\sim$	10^4 cm
ℓ = Small eddy size	$\sim$	1 cm

Here ℓ is the scale on which energy dissipation by viscosity occurs; in water the kinematic viscosity $\nu \approx .01\ cm^2/s$. The Reynolds number is $R = \overline{v}L/\nu \approx 1.5\ 10^8$. The inertial range spectrum of this flow is plotted in Figure 1, where $\phi_1(k)$ the one-dimensional spectrum (Batchelor 1953),[1] ε is the rate of energy dissipation, and $k_d = (\varepsilon/\nu^3)^{1/4}$ is the Kolmogorov dissipation wavenumber. Notice that the inertial range spectrum $\phi_1(k) \propto k^{-5/3}$ extends over two decades of wavenumber.

In order to simulate this flow accurately over all dynamically important scales, it is necessary to resolve the range of scales 10^4 to 1, so that at least 10^4 degrees of freedom are required in each of three space directions. Consequently, about $(10^4)^3 = 10^{12}$ degrees of freedom (velocity, pressure values, etc.) must be retained. To be useful, the

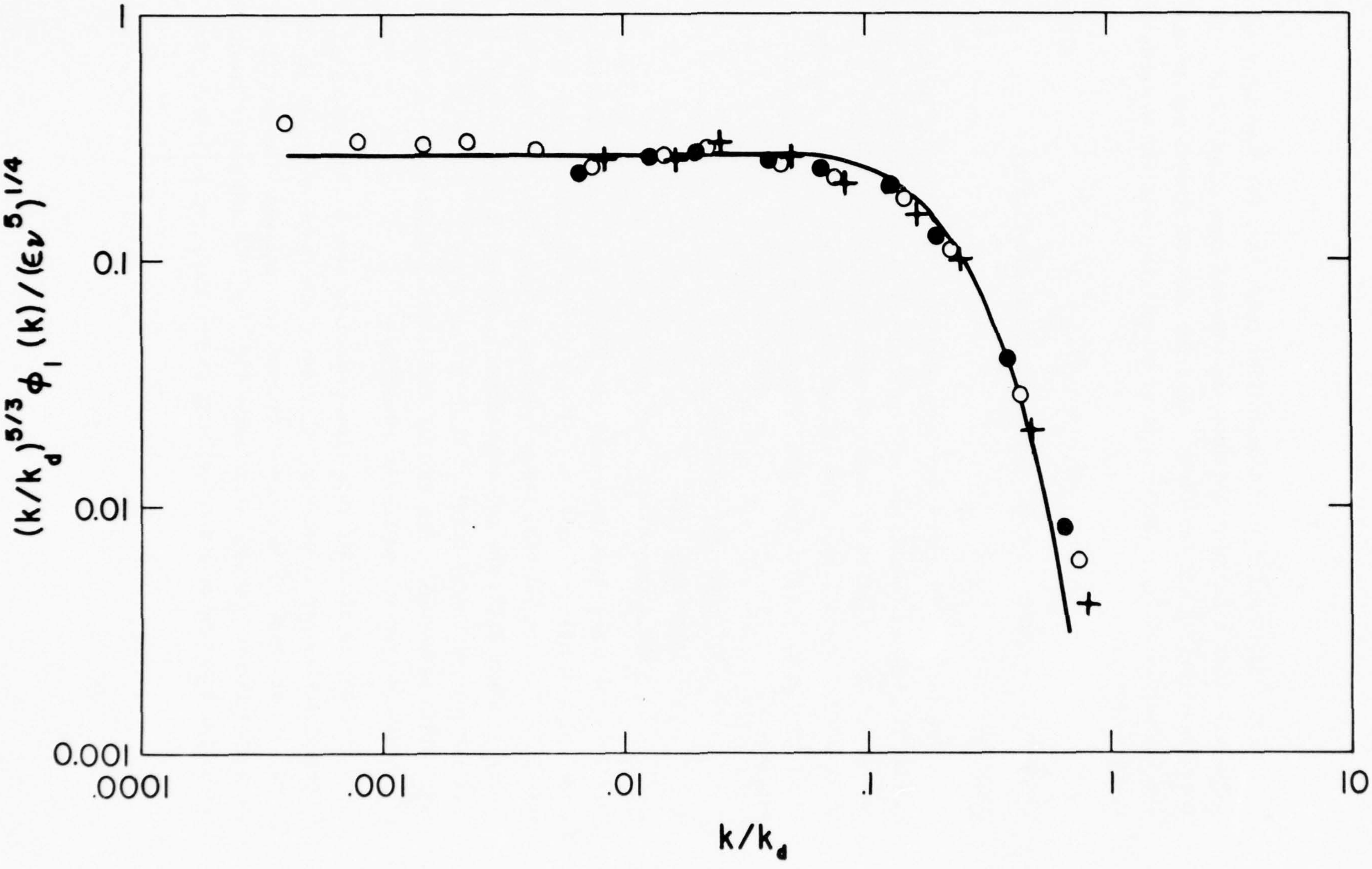


Figure 1.

calculation of (1) should proceed for some dynamically significant time, typically the large eddy circulation time, which is about $L/\bar{v} \approx 60s$. However, numerical solution of (1) must proceed in small time steps, both to maintain accuracy and numerical stability, typically of the size $\ell/\bar{v} \approx 6\times10^{-3}s$. Consequently, about 10^4 time steps are required to calculate one eddy circulation time. A typical numerical solution of (1) requires about 100 computer operations (instructions) per time step per retained degree freedom. Therefore, with 10^4 time steps and 10^{12} degrees of freedom, about 10^{18} operations are required to complete the simulation. Even with the next generation machines that may operate at 1 ns($=10^{-9}$s) per operation at a cost of about \$1/s, about $10^9 s \simeq 30$ years and \$$10^9$ are required to complete a single computation!

The estimates just given may be formalized as follows: the large scale L of a turbulent flow is the size of the energy contained eddy motions and is typically related to the size of the body or scale of the forces generating the motion. The turbulent motions on smaller scales give rise to enhanced rates of transport in the flow over corresponding molecular transfer rates, including enhanced energy dissipation, momentum transfer, heat transfer, and particle diffusion. These enhanced transport rates are often modelled by eddy transfer coefficients which turn out to be many orders of magnitude larger than molecular coefficients when the Reynolds number R is large.

It has been conjectured [with some experimental and theoretical support (cf. Batchelor 1953[1] and Orszag 1975[18])] that the energy dissipation rate ε remains $O(1)$ as $R \to \infty$ in a turbulent flow, in contrast to the elementary estimate $\varepsilon = O(1/R)$ in a laminar flow. This property that $\varepsilon = O(1)$ as $R \to \infty$ is intimately related to the idea that the small scales in a turbulent flow adjust themselves to provide the required enhanced transport as $R \to \infty$. In the case of energy dissipation, this behavior requires that the dissipation scale ℓ, which is the smallest dynamically important scale in the flow, must be of order

$$\ell = (\nu^3/\varepsilon)^{1/4} \simeq L/R^{3/4} \quad \text{as } R \to \infty \tag{10}$$

Equation (10) follows from dimensional analysis: $[\ell] = \text{cm}$, $[\varepsilon] = \text{cm}^2\text{s}^{-3}$, $[\nu] = \text{cm}^2\text{s}^{-1}$, while ℓ can depend only on ε and since:(a) in homogeneous turbulence, ℓ adjusts itself to maintain $\varepsilon = O(1)$ because ε provides the only dynamical coupling of small scales to the large energy containing eddies; (b) the precise scale of dissipation is governed by the molecular viscosity ν which must do the actual dissipation of energy. A similar argument implies that, in the inertial range, which consist of small eddies that are sufficiently large that molecular viscosity ν has no direct effect on them, the energy spectrum $E(k)$ must satisfy Kolmogorov's law

$$E(k) = C\,\varepsilon^{2/3}\,k^{-5/3} \; ; \tag{11}$$

(11) also follows by dimensional analysis since $[E(k)] = \text{cm}^3\text{s}^{-2}$ and $[k] = \text{cm}^{-1}$ while $E(k)$ can depend only on ε and k, the wavenumber, in the inertial range. The qualitative spectrum of turbulence predicted by this dimensional theory is plotted in Figure 2.

It follows from (10) that the range of spatial scales that must be accurately resolved in a calculation at Reynolds number R is of order

$$L/\ell \simeq R^{3/4},$$

so that the total number of spatial degrees of freedom is of order $(R^{3/4})^3 = R^{9/4}$. The calculation must proceed for a time of order $L/\overline{v}$, while time steps must be restricted to order $\ell/\overline{v}$, so that order $L/\ell \simeq R^{3/4}$ time steps must be taken per simulation run. Consequently, the total number of operations scales as $R^{9/4}\,R^{3/4} = R^3$.

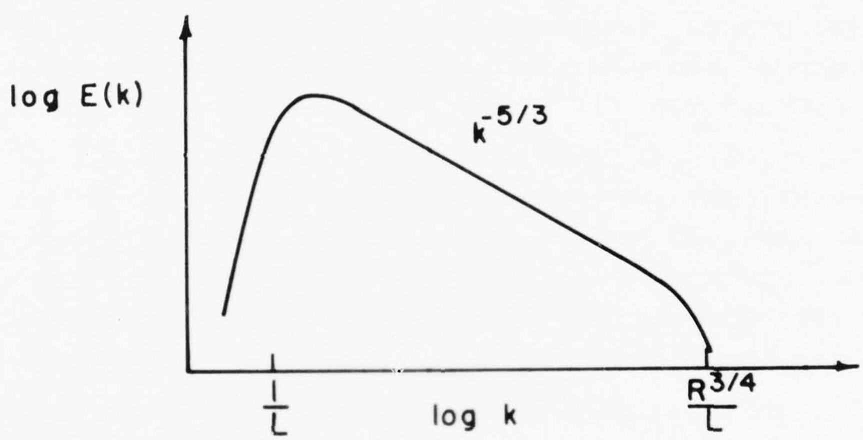


Figure 2.

The discouraging result of this argument is that every increase of computer power by a factor 10 allows only an increase by a factor $10^{1/3} \approx 2.15$ in the Reynolds number.

Estimates like those just given have seemed to spell doom upon all reasonable attempts at numerical solution of the turbulence problem. However, it is unreasonable to be so pessimistic. If a calculation at $R = 10^8$ requires 10^{21} operations (which is a better estimate than the 10^{18} obtained above) then the R^3 dependence implies that a calculation at $R = 10^4$ requires only 10^9 operations or 15 minutes on a 1 MIPS machine. There are many interesting turbulent flows at Reynolds numbers $R \sim 10^4$-10^5. In fact, most laboratory turbulence experiments fall within this range -- only geophysical environments provide tractable sources of data at much higher Reynolds numbers.

The numerical simulations of turbulence reviewed in the next section have been performed at Reynolds numbers in the range 10^3-10^5. At these Reynolds numbers, the computer experiments can complement and supplement laboratory experimental data. The question of the relevance of these moderate Reynolds number turbulence experiments to huge Reynolds number geophysical flows is explored further in <u>COMPARISON WITH OTHER METHODS</u>.

There are two important practical difficulties with numerical simulations of turbulence in addition to the resolution problem outlined above for homogeneous turbulence; these additional difficulties also relate to inadequate resolution but are of a somewhat difference nature then the difficulty discussed above. First, there is the difficulty with obtaining adequate statistics to compute accurate statistical averages and, second, there are a variety of difficulties relating to the imposition of boundary conditions on the flow.

The difficulty with statistics owes to the fact that the arithmetical mean of N independent values of a random variable with average A and standard deviation D fluctuates about A with amplitude of order $D/\sqrt{N}$.

Consequently, increasing the sample size by a factor 100 only decreases errors by a factor 10. Large samples are necessary for high accuracy.

Large samples are readily available for homogeneous turbulence, since three-dimensional spatial averages may be used. In the isotropic turbulence simulations discussed in the next section, averages were obtained as arithmetical means over bands in wave space, i.e. as averages over all wavevectors satisfying $k-\frac{1}{2}\Delta k<|\vec{k}|<k+\frac{1}{2}\Delta k$, where the bandwidth was chosen sufficiently large that most spectra so obtained were accurate to better than 10%. On the other hand, in shear flows, sample size is a serious problem. In two-dimensional shear flows, averages are functions of x (the downstream direction) and z (the cross-stream direction), but not of y (the spanwise direction). In this case, spatial averages may be obtained as averages over the y-direction; if the flow is longitudinally homogeneous (statistically homogeneous in x) then the x-direction may also be used for averaging. Similarly, in a statistically axisymmetric shear flow, the azimuthal direction and perhaps the longitudinal direction may be used for spatial averages. However, realistic shear flows have at least one direction of inhomogeneity and this means a significant degradation of the accuracy of spatial statistics compared with corresponding statistically homogeneous flows.

In a general turbulent flow, the situation with regard to statistics is very serious. If there is no direction of spatial homogeneity, then averages may be computed in but two ways. First, time averages may be used <u>if</u> the flow is statistically stationary. However, it is not sufficient to use the flow values at N successive time steps to get N independent measurements, because the flow is strongly correlated from one time step to the next. In order to get N independent samplings of an energy-containing scale of motion, it is necessary to calculate through order N large-eddy circulation times $L/\bar{v}$. This increases our previous operation estimates by a factor N.

Second, averages of an arbitrary (statistically inhomogeneous, non-stationary) flow can always be computed as an ensemble average, in which the average is taken over N distinct flows with the same statistics. However, computing an ensemble average increases the amount of work by a factor N over that necessary to compute a single flow.

The final difficulty with numerical simulation of turbulence that we wish to discuss concerns boundary conditions. In Figure 3, we give a schematic representation of the computational region for turbulent flow past a body. There are three general regions where boundary conditions must be imposed: at the inflow boundary, at the outflow boundary, and on the body.

Inflow boundaries: It is not possible to deal directly with infinite spatial regions because of the finite capacity of computers. There are two possible procedures for dealing with this problem. One way is to transform the infinite region into a finite region by use of a map, like $z=x/(x+1)$ which maps $0\leq x<\infty$ into $0\leq z<1$. As explained by Orszag et al (1975),[20] the use of such maps requires considerable care; mapping is successful only if the asymptotic behavior of the solution being sought is sufficiently simple. In the case of flow past a body, the downstream outflow is not at all simple (since it comprises the turbulent wake region), so mapping does not seem to hold promise there. However, the flow at the upstream boundary may be 'simple' and mapping may be appropriate, though to the author's knowledge it has not yet been tried in this situation. The other method is to truncate the computational domain at some finite distance, as shown in Figure 3. In this case, the inflow velocity field must be completely specified. The arbitrariness of this specification influences the rest of the computation. It may seem that the truncation method involves more arbitrariness than mappings do. However, this is not so, as it may be shown (Orszag et al (1975))[20] that both procedures have similar properties. In summary, the

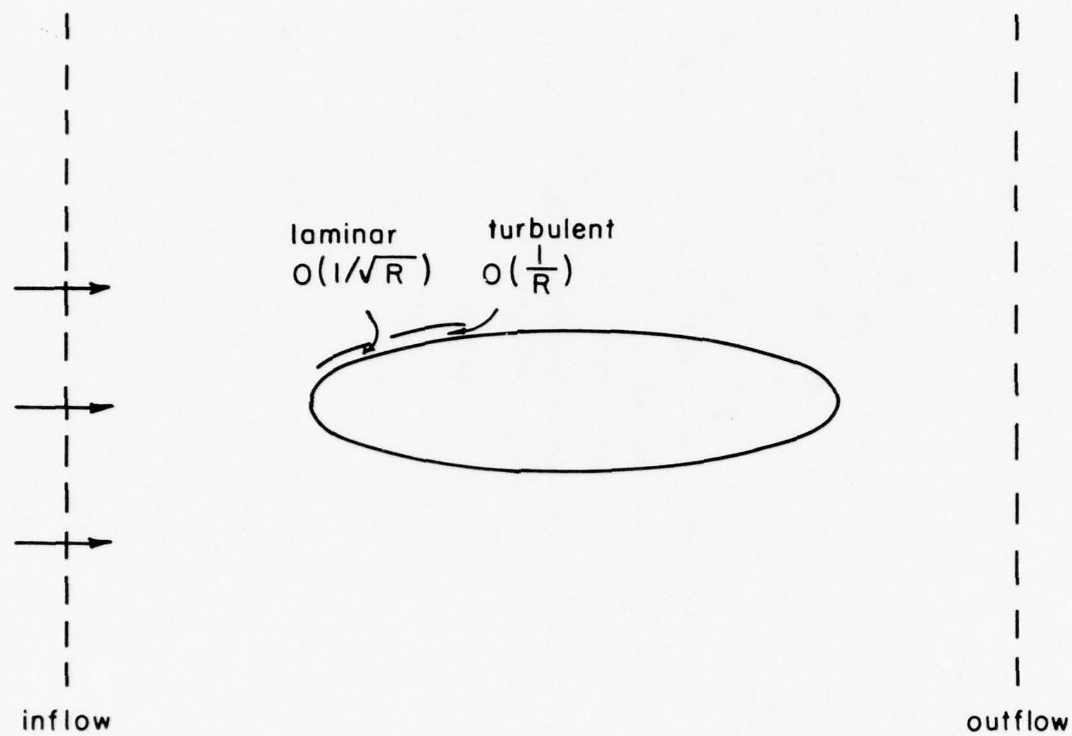


Figure 3.

difficulty with upstream (inflow) boundaries concerns the lack of knowledge of the required upstream flow. A similar arbitrariness appears in the specification of initial conditions in turbulence simulations. Presumably, this lack of knowledge does not affect the final results strongly, in the sense that while individual realizations of turbulence are highly sensitive and unstable, average properties are strongly stable to perturbations.

Outflow boundaries: The situation concerning outflow boundary conditions is potentially very serious. Here it seems physically unrealistic to permit specification of the complete flow field. Yet, viscous flow theory shows that the complete velocity field should be specified at an outflow boundary; inviscid flow theory suggests that specification of either the normal component of the outflow velocity or the pressure is sufficient to set the problem. However, all these flow features are *a priori* unknown; in fact, it is certain that a large obstacle, say, placed some distance downstream of the outflow boundary will affect the flow within the computational domain. In a turbulent flow, it seems that the only hope is that specific details of downstream boundary conditions have little effect upstream. There is some evidence for this behavior in simulations of boundary layer transition but it remains to be verified. The real danger with downstream boundaries is that by overspecification the flow may be modified in a very important way. For example, if both the normal component of velocity and the pressure are given at outflow, then the drag coefficient of the body is completely specified, most likely in error compared with the true drag coefficient. Drag should be a result of the computation, not something unwittingly imposed during problem formulation.

Boundary conditions on the body: The difficulty here shows itself in several ways. If there is a laminar boundary layer on the body, its thickness is order $R^{-1/2}$ compared with the body size. At large R, it seems that high resolution is required to resolve the boundary layer.

However this is not difficult to achieve, even at high Reynolds number, if a mapping is used to stretch the coordinate normal to the body. This mapping solves the problem if the length scale of the turbulence exterior to the boundary layer is much larger than the boundary layer thickness. Because the boundary layer is laminar, only low resoltuion is required streamwise along the boundary layer. On the other hand, if the boundary layer is turbulent, the viscous sublayer thickness is order 1/R. Again the direction normal to the boundary can be handled by transformation (if turbulent length scales outside the boundary layer are much larger than the sublayer thickness). However, the principal difficulty with a turbulent boundary layer or a laminar one undergoing transition to turbulence is that the streamwise direction must also be resolved. Since typical streamwise length scales are of the same order as the boundary layer thickness (as for a turbulent boundary layer) or at most an order of magnitude larger (as for Tollmien-Schlichting waves in a boundary layer undergoing transition), it follows that streamwise resolution must be order 1/R in the boundary layer. This restriction is exceedingly severe and will hinder any attempt to do a good job of simulating turbulent flow about realistic body shapes.

A possible way around the difficulty just raised is to model the turbulent boundary layer by invoking conditions on wall stress or other similar quantity. There is no general theory concerning this kind of turbulence modelling, though some recent attempts have been moderately successful (e.g. Deardorff 1970).[2,3] We do not enter into a discussion of these questions here because it falls within the general realm of turbulence modelling (COMPARISON OF OTHER METHODS), not direct solution of the Navier-Stokes equations.

SURVEY OF APPLICATIONS

Operational computer codes for the numerical simulation of turbulence include codes capable of using up to 65,000 degrees of

freedom to represent each of the dynamical variables, including velocity, temperature, pressure, etc. Two-dimensional homogeneous turbulence codes are operational with up to 256×256 modes [cutoff $K = 128$ in (5), (6)], while three-dimentional shear and homogeneous turbulence codes are running with 32×32×32 modes ($K = 16$), 64×8×128 modes, etc. For example, a 32×32×32 three-dimentional turbulence calculation can be used to simulate wind tunnel turbulence at a Reynolds number of order 25,000 and requires about 3 s per time step and several hundred time steps on a CDC 7600.

The most elementary application of numerical methods to gain information of interest about homogeneous turbulence is the Taylor-Green vortex (Taylor and Green 1937). Here the initial velocity field is nonrandom

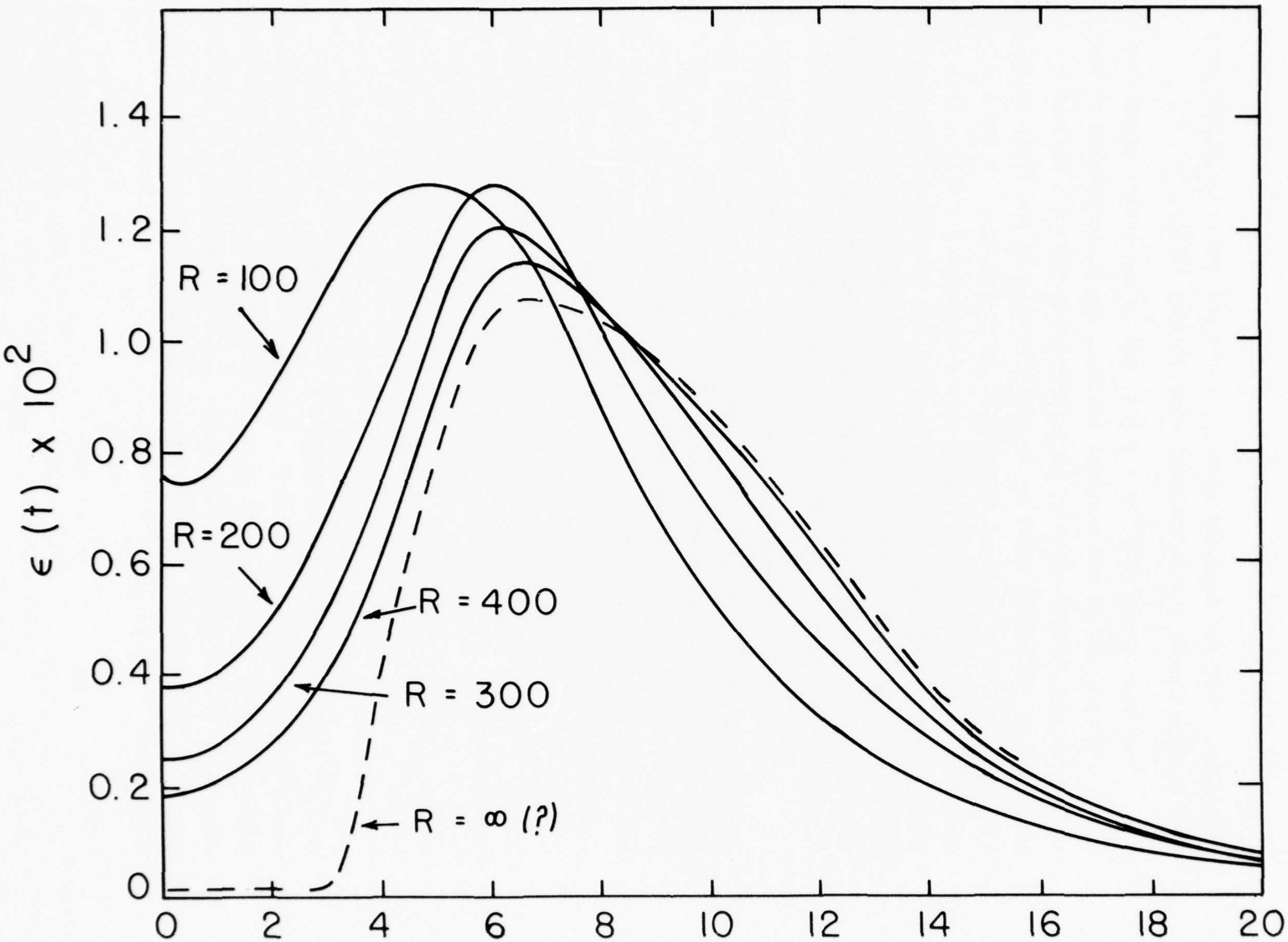
$$\begin{aligned} v_1(\vec{x},0) &= \cos x_1 \sin x_2 \cos x_3 \\ v_2(\vec{x},0) &= \sin x_1 \cos x_2 \cos x_3 \\ v_3(\vec{x},0) &= 0 \end{aligned} \tag{12}$$

for which the vortex lines are twisted. The flow field does not remain two-dimensional for $t > 0$ and the vortex lines are stretched by the self-induced shear. Enhancement of vorticity by stretching of vortex lines in a local shear flow is the fundamental mechanism involved in the enhanced energy dissipation of turbulence relative to laminar flow that is the basis of the Kolmogorov theory of turbulence (previous section). In fact,

$$\varepsilon = \nu \overline{\omega^2} \tag{13}$$

where $\vec{\omega}$ is the vorticity; (13) shows directly how vortex stretching by convection can enhance energy dissipation.

Numerical solution of the Navier-Stokes equations with the initial conditions (11) gives the results shown in Figure 4 for the dissipation



1.4
1.2
1.0
0.8
0.6
0.4
0.2
0
ϵ (t) x 10^2
R = 100
R = 200
R = 400
R = 300
R = ∞ (?)
0
2
4
6
8
10
12
14
16
18
20

Figure 4.

rate $\varepsilon(t)$. Here the Reynolds number $R = 1/\nu$ and the calculations were done using a cutoff $K = 16$ spectral code (Orszag 1974b).[17]

The increase of $\varepsilon(t)$ for $t \lesssim 5$ is due to the vortex stretching mechanism. As remarked in the previous Section, the basic premise of the Kolmogorov and related theories of turbulence is that $\varepsilon = O(1)$ as $R \to \infty$. According to (13), this behavior is possible only if the limiting behavior of $\overline{\omega^2}$ as $\nu \to 0$ $(R \to \infty)$ is as follows: $\overline{\omega^2}$ is finite as $\nu \to 0$ for $t < t_\star$, while $\overline{\omega^2} \to \infty$ as $\nu \to 0$ for $t > t_\star$ in such a way that $\varepsilon = O(1)$. This hypothetical behavior is plotted as the curve $R = \infty$ in Figure 4 (Orszag 1974b).[17] Admittedly, the Taylor-Green results shown in this figure are far from conclusive that $\varepsilon = O(1)$ as $\nu \to 0$, but there is scant additional theoretical information on which to test the hypothesis, first inferred experimentally.

Some results of numerical simulations of two-dimensional turbulence are given in Figures 5 to 8. In Figure 5, we plot four enstrophy (mean-square vorticity) dissipation spectra $k^4E(k)$ vs k (Herring et al 1974).[7] In two dimensions, vortex lines cannot be stretched, so $\overline{\omega^2}(t) \leq \overline{\omega^2}(0)$ and energy dissipation $\varepsilon(t) \leq \varepsilon(0) = O(\nu)$ as $\nu \to 0$ (cf. Orszag 1975).[18] Therefore, Kolomogorov's dimensional reasoning invoked in the previous Section must be reconsidered. In fact, while $\varepsilon(t)$ decreases with t in two dimensions, the rate of enstrophy dissipation $\eta(t)$ may be increased by shear. Here

$$\eta(t) = -\frac{d}{dt}\overline{\omega^2} = 2\nu \int_0^\infty k^4E(k)dk \tag{15}$$

so $2\nu k^4E(k)$ is the spectrum of enstrophy dissipation. It turns out that this new quantity $\eta(t)$ plays a dynamical role in two dimensions similar to that played by $\varepsilon(t)$ in three dimensions. In fact, accurate resolution of the enstrophy dissipation spectrum guarantees an accurate numerical simulation of the dynamically important scales of two-dimensional turbulence.

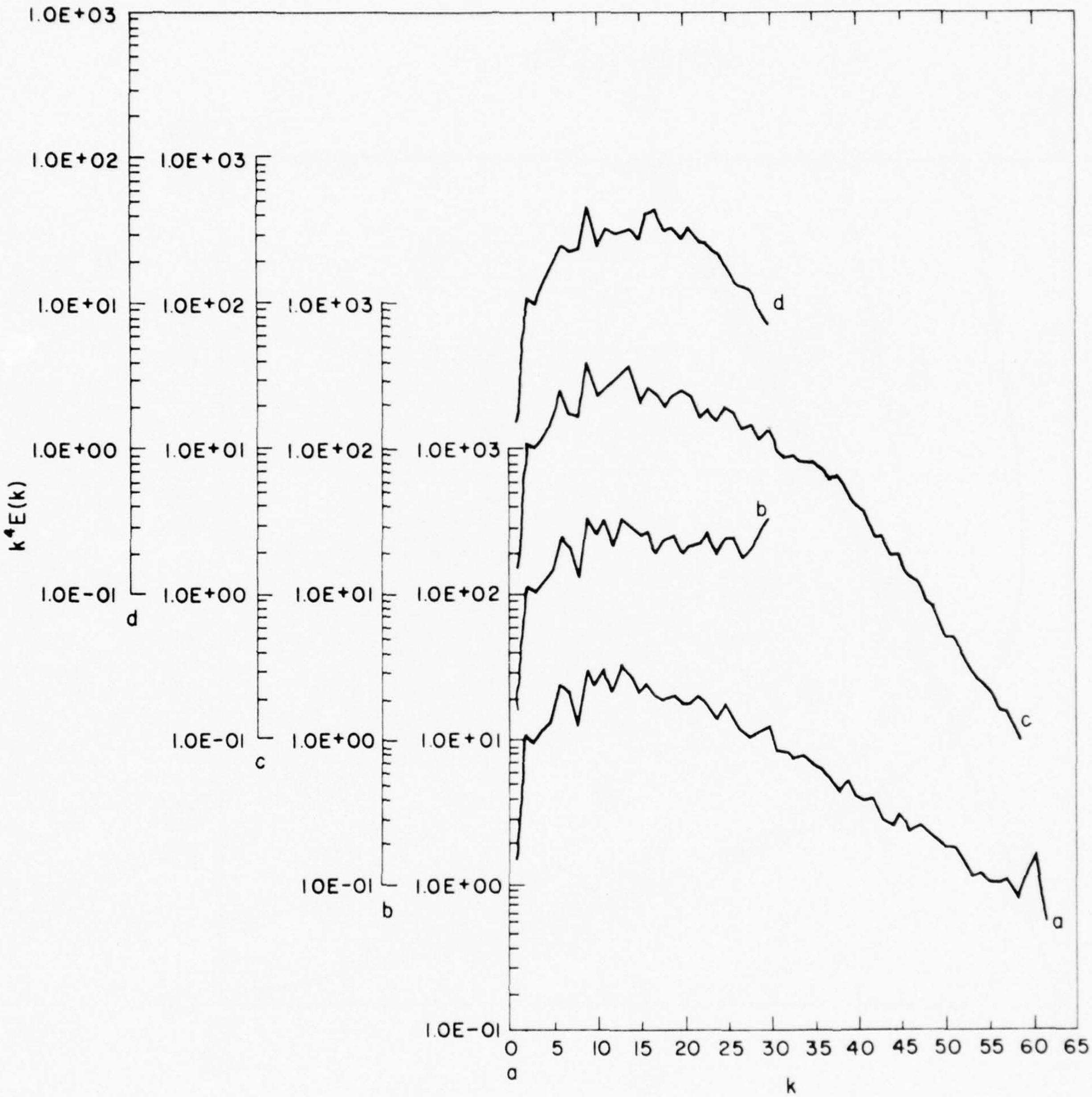

1.0E+03
1.0E+02
1.0E+01
1.0E+00
1.0E-01
d
1.0E+03
1.0E+02
1.0E+01
1.0E+00
1.0E-01
c
1.0E+03
1.0E+02
1.0E+01
1.0E+00
1.0E-01
b
1.0E+03
1.0E+02
1.0E+01
1.0E+00
1.0E-01
a
$k^4E(k)$
0 5 10 15 20 25 30 35 40 45 50 55 60 65
k
a
b
c
d

Figure 5.

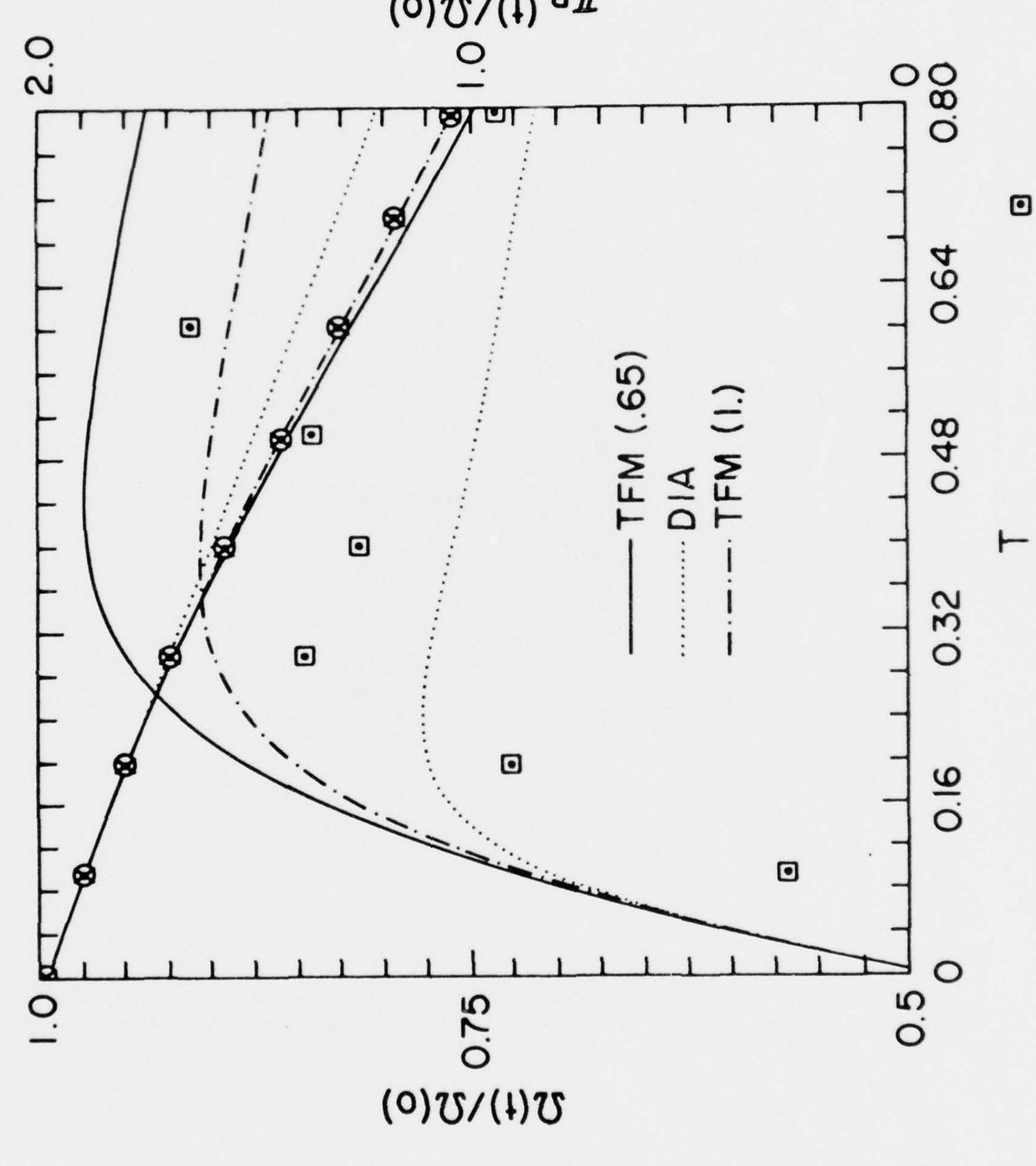
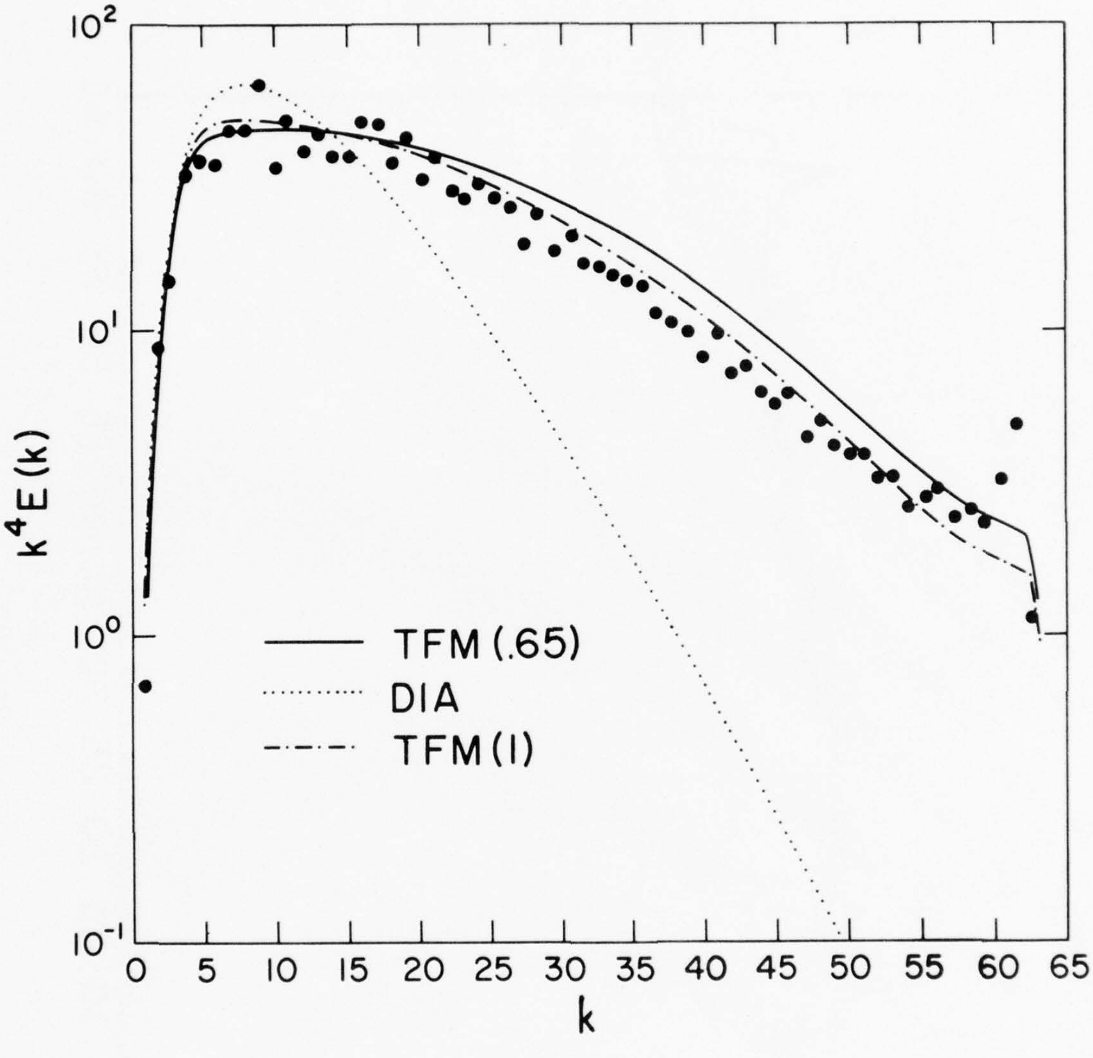


Figure 6.



10^2
10^1
10^0
10^{-1}
$k^4E(k)$
TFM (.65)
DIA
TFM (1)
0 5 10 15 20 25 30 35 40 45 50 55 60 65
k

Figure 7.

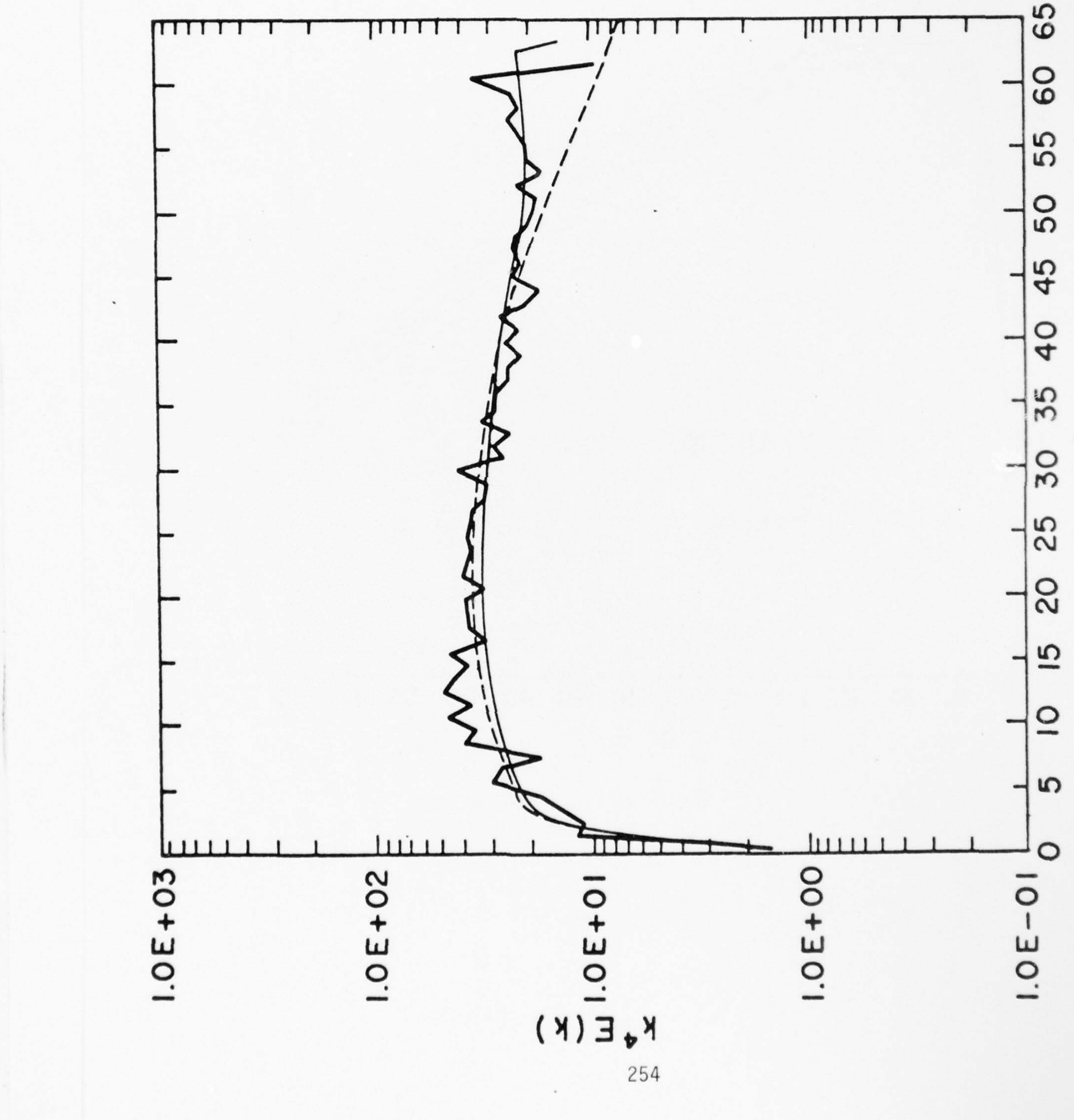

1.0E+03
1.0E+02
1.0E+01
1.0E+00
1.0E-01
$k^4E(k)$
0
5
10
15
20
25
30
35
40
45
50
55
60
65
k

Figure 8.

The enstrophy dissipation spectra in Figure 5 are obtained by solution of the Navier-Stokes equations from the same initial conditions out to $t = 2$ (a significant time of evolution) using four different numerical methods: (a) 128×128 (cutoff $K = 64$) spectral method; (b) 64×64 spectral method; (c) 128×128 finite-difference method; (d) 64×64 finite difference method. The Reynolds number of these simulations is roughly 350 based on integral scale and rms velocity. Figure 5 illustrates several attractive features of the spectral calculations already discussed in METHODS. First, comparison of Figure 5(a) with Figure 5(b) shows that increasing the spectral resolution from 64×64 to 128×128 does not affect wavenumbers $k \lesssim 15$, while similar comparison of Figures 5(c), 5(d) suggests that increasing resolution of the finite difference codes does affect the results for $k \gtrsim 3$. On the assumption that the 128×128 spectral results are accurate at all scales (and they may be shown to be very nearly so), it is apparent that the 64×64 spectral results are comparably accurate as the 128×128 finite difference results, even at the crude accuracy level of comparing graphs.

Second, Figure 5 illustrates the bootstrap capability of the spectral calculations. For the 64×64 spectral results show that $\eta(t)$ is only marginally resolved so that the 64×64 spectral results are not accurate at all sacles. The 128×128 spectral results show that $\eta(t)$ is adequately resolved and that the results are accurate at all scales. In contrast, the finite difference results shown in Figures 5(c), 5(d) seem to show that $\eta(t)$ is adequately resolved since the spectrum decreases rapidly to large k; yet neither calculation is accurate at all scales.

Figures 6 and 7 present some further results of the calculations used to construct Figure 5. In Figure 6, there is plotted the mean-square vorticity $\Omega(t) = \overline{\omega^2(t)}$ vs t for the 128×128 spectral calculation (circled dots) and for several analytical theories of turbulence (curves).

The curves labelled $\Pi_B(t)$ will not be discussed here (cf. Herring et al 1974).[7] In Figure 7, a similar comparison is made between the 128×128 spectral results (points) for $k^4E(k)$ and the theories (curves). The important result of Figures 6 and 7 for the present discussion is the illustration that it is possible to obtain comparison between theory and numerical experiment; these comparisons have proven to be extremely useful in understanding the limitations of turbulence theories and in generating ideas for their improvement.

Another basic question concerns the possibility for verifying theories of the inertial range by direct numerical simulation. In two dimensions, dimensional analysis similar to Kolmogorov's (Kraichnan 1967),[10] implies that there is a two-dimensional range with

$$E(k) = C'\eta^{2/3} k^{-3} \tag{16}$$

with C' a constant. It does not seem possible to give definitive test of (16) with only 128×128 spectral resolutions (Herring et al 1974);[7] it is hoped that new 512×512 spectral calculations will be more useful in this regard. Some results from high Reynolds number ($R \simeq 10^3$) 128×128 computations are plotted in Figure 8. The heavy solid curve is the result of the Navier-Stokes calculations for $k^4E(k)$; rather than (16) it seems that there is a broad spectral range over which $E(k)\propto k^{-4}$, as predicted for the inertial range by Saffman (1971).[25] However, this conclusion is premature, as shown by the other two curves in Figure 8. The dashed line is the result of numerical solution of the integro-differential equations of the test-field model, an analytical turbulence theory (Kraichnan 1971)[11] using a wavenumber cutoff K = 128; the light solid curve is the result of a similar calculation with K = 64, the same cutoff used to obtain the heavy solid curve. The conclusion from these comparisons is that the apparent k^{-4} spectral range is due to the spectral cutoff, not necessarily the basic physics of the turbulence

problem. In fact, it is known analytically that the test-field model, which is in good agreement with the numerical simulations of two-dimensional turbulence as shown by Figures 6 - 8, yields a k^{-3} inertial range behavior.

Some results of numerical simulations of three dimensional homogeneous turbulence are shown in Figure 9 (Orszag and Patterson 1972).[22] Again, the results of the simulations are compared with those of another analytical turbulence theory, the direct-interaction approximation (Kraichnan 1959).[8] These simulations were performed at wind-tunnel-like Reynolds numbers (35, based on the Taylor microscale; roughly 15,000, based on mesh separation and air speed in the corresponding wind tunnel).

Additional results of three dimensional turbulence simulations are shown in Figures 10, 11. Here the evolution of two velocity fields that differ from each other initially by only a small perturbation is studied, a problem of interest with regard to the instability of turbulence with important applications to the predictability of atmospheric motions (Leith 1971).[14] In Figures 10(a), (b), the kinetic energy contours of the two initial velocity fields are plotted. In Figures 11(a), (b), the corresponding contours after evolution for about one large eddy circulation time are plotted; a $32 \times 32 \times 32$ spectral code was used to obtain these results. It is striking how the relatively small initial perturbation grows into a large perturbation during time evolution.

Numerical simulations of turbulent shear flows by direct solution of the Navier-Stokes equations have been pursued only very recently. Orszag and Pao (1974)[21] report the results of simulations of the wake of a self-propelled body. Because of the general difficulties with inhomogeneous turbulence simulations (especially statistics), these calculations were performed by isolating a slab of the wake and following its downstream evolution by Taylor's hypothesis.

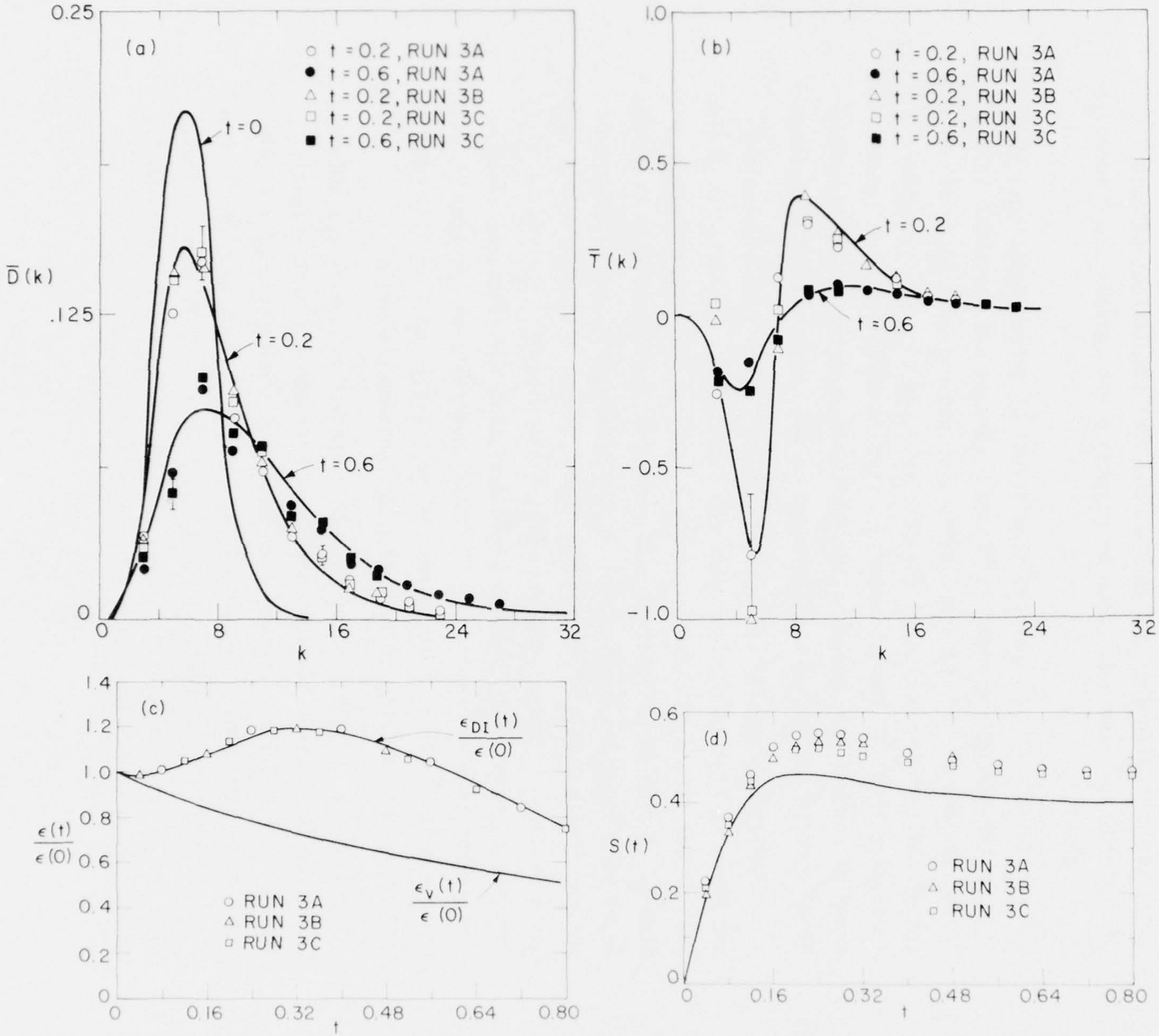
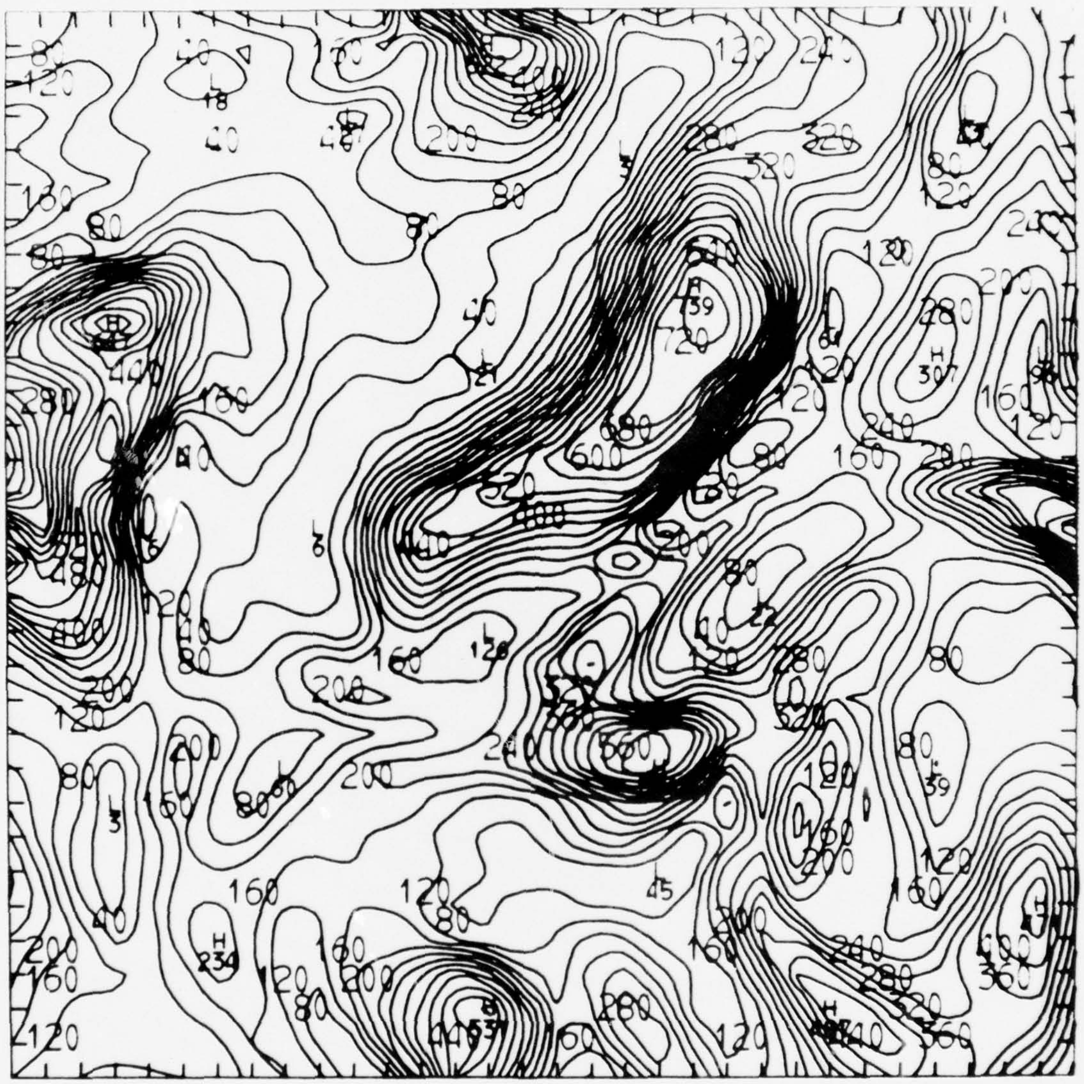
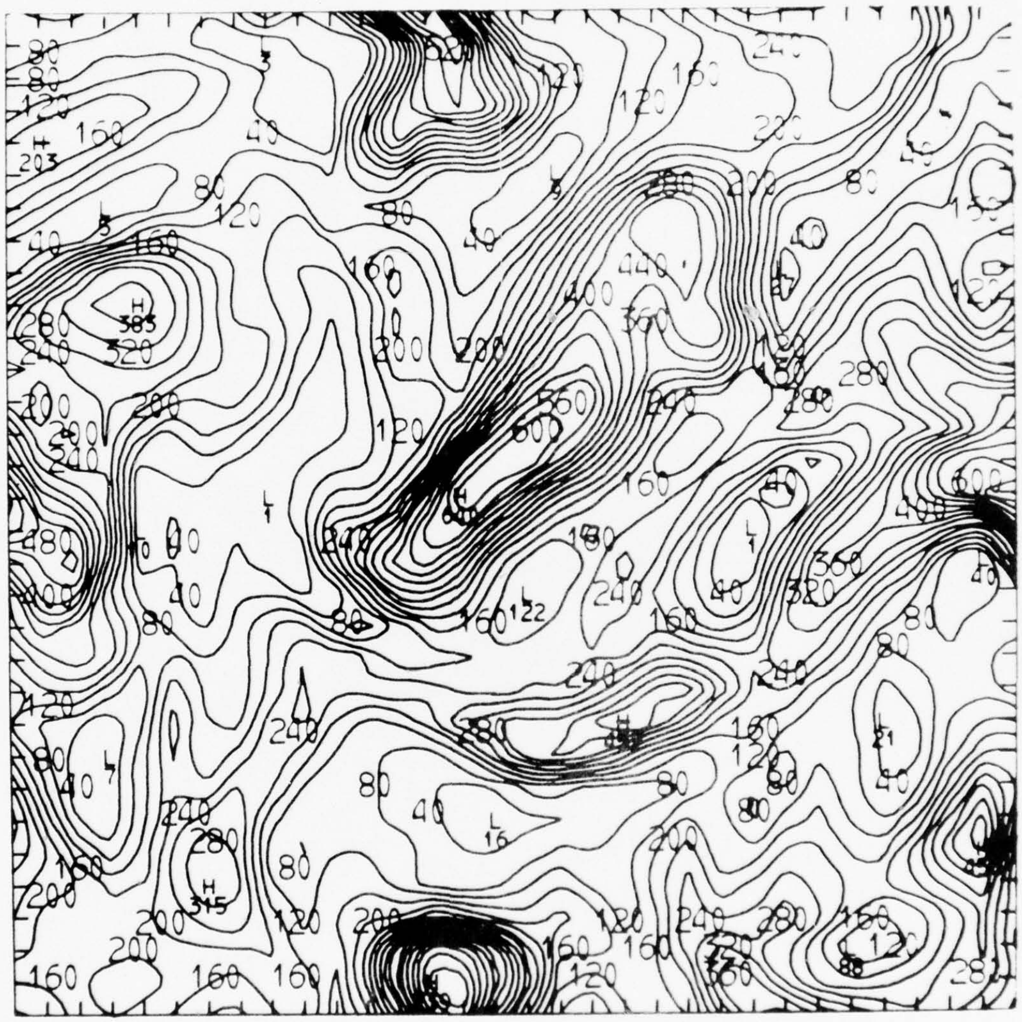

(a)
t = 0.2, RUN 3A
t = 0.6, RUN 3A
t = 0.2, RUN 3B
t = 0.2, RUN 3C
t = 0.6, RUN 3C
t = 0
t = 0.2
t = 0.6
$\bar{D}(k)$
k
(b)
$\bar{T}(k)$
(c)
$\frac{\epsilon_{DI}(t)}{\epsilon(0)}$
$\frac{\epsilon_v(t)}{\epsilon(0)}$
$\frac{\epsilon(t)}{\epsilon(0)}$
RUN 3A
RUN 3B
RUN 3C
t
(d)
S(t)

Figure 9.



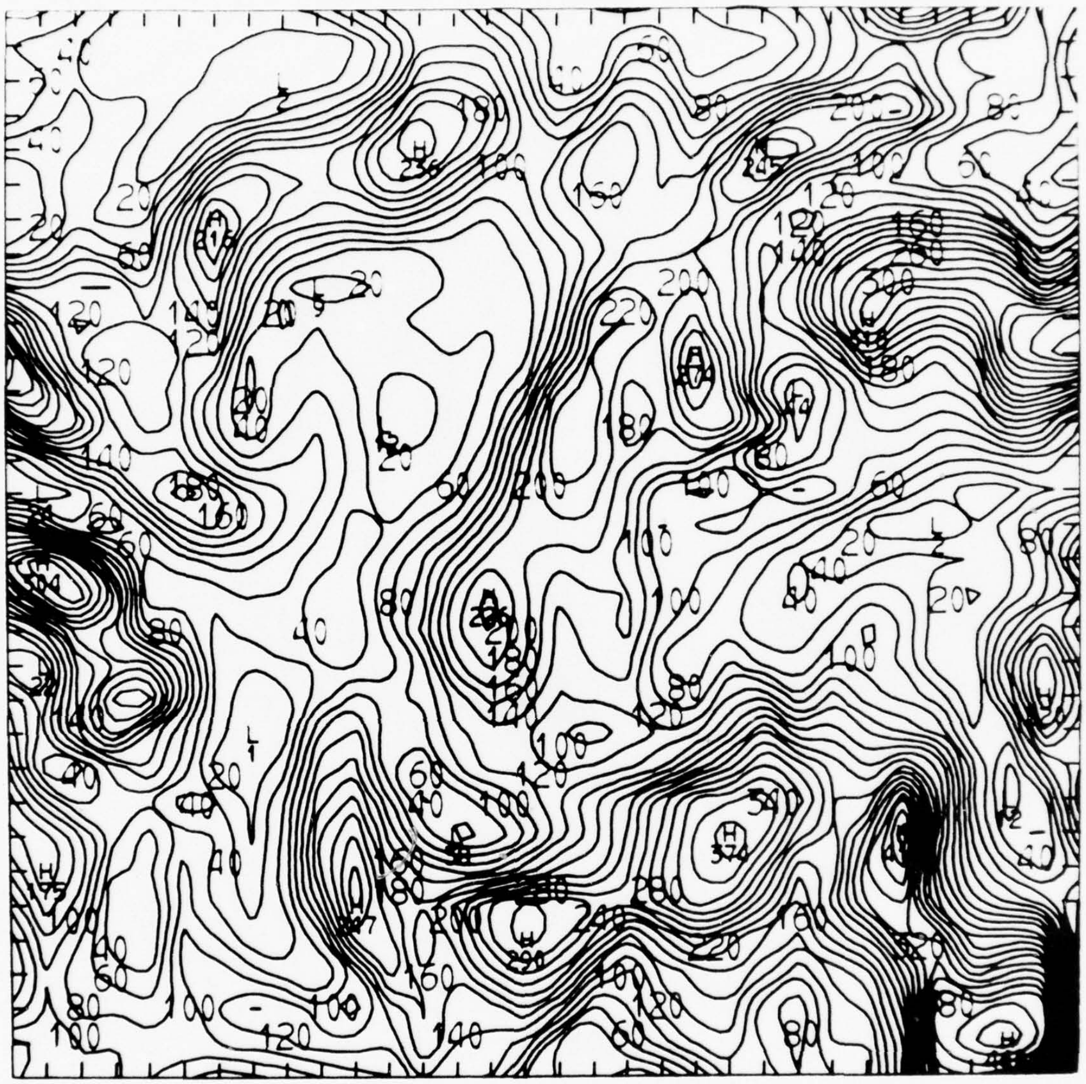
CONTOUR FROM 0.0 TO 7.200E+00 CONTOUR INTERVAL OF 4.000E-01 SCALED BY 1E+02 PT(5,5)= 1.066E+00

Figure 10a.



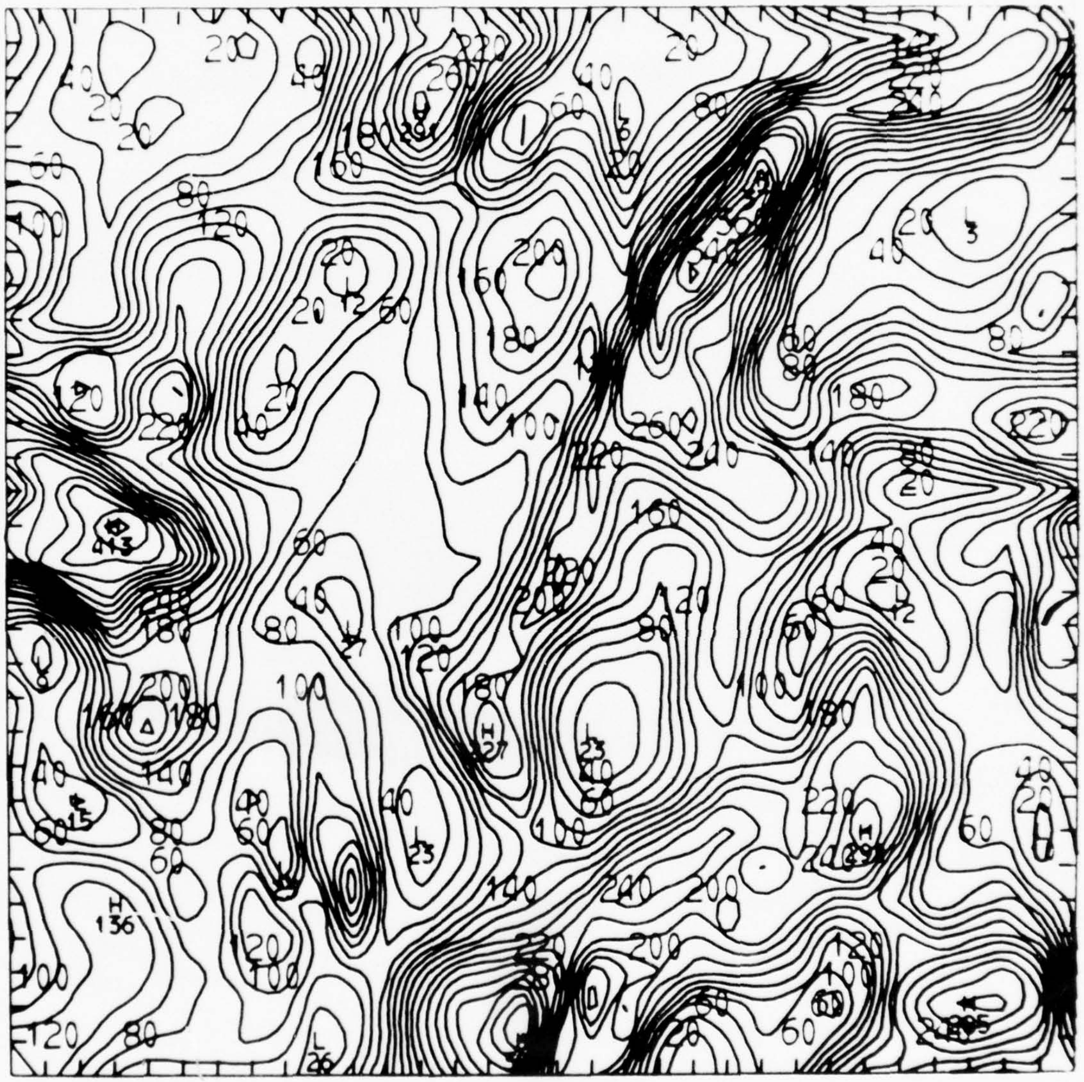
CONTOUR FROM 0.0 TO 7.200E+00 CONTOUR INTERVAL OF 4.000E-01 SCALED BY 1E+02 PT(3,3)= 1.742E+00

Figure 10b.



CONTOUR FROM 0.0 TO 4.200E+00 CONTOUR INTERVAL OF 2.000E-01 SCALED BY 1E+02 PT(3,3)= 7.479E-01

Figure 11a.



CONTOUR FROM 0.0 TO 4.000E+00 CONTOUR INTERVAL OF 2.000E-01 SCALED BY 1E+02 PT(5,5)= 1.204E+00

Figure 11b.

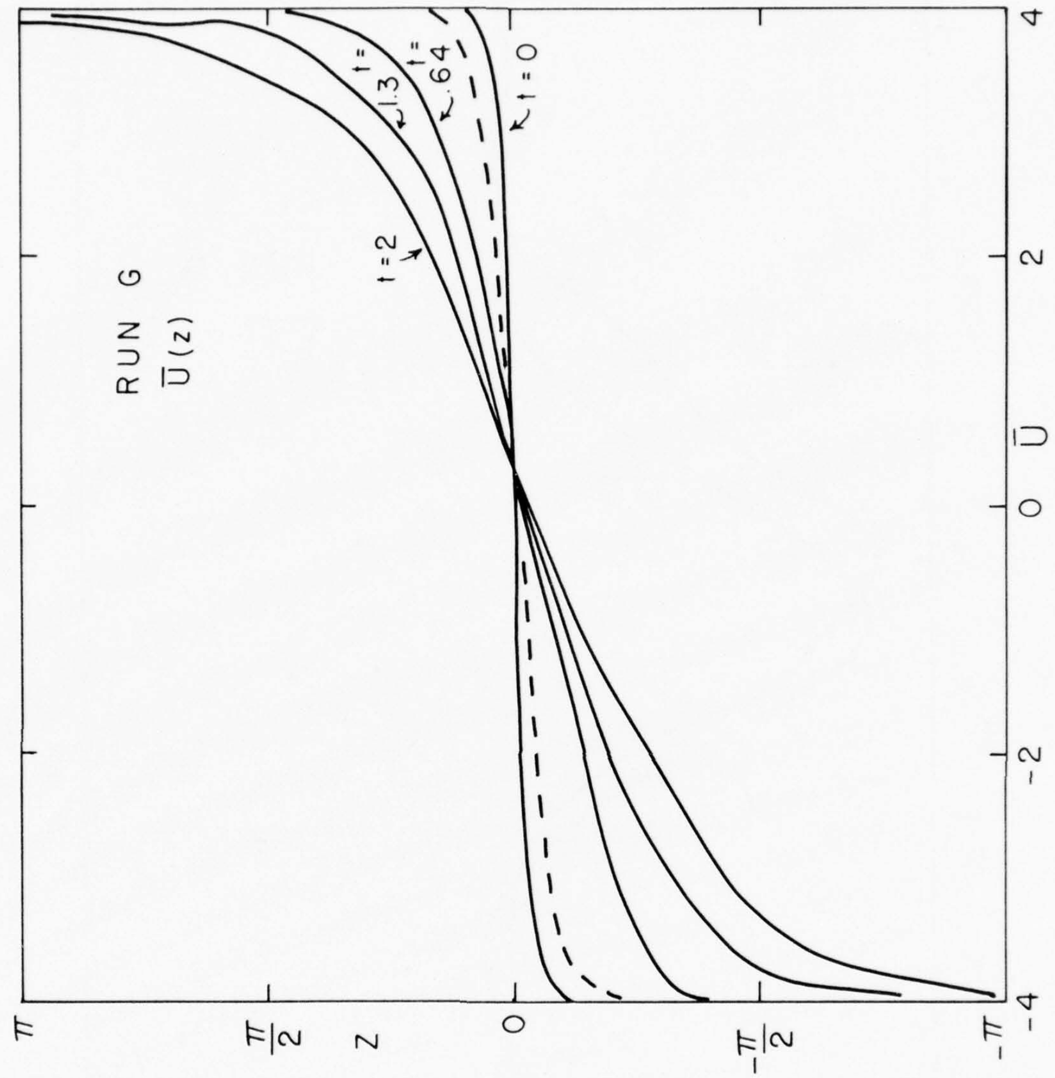
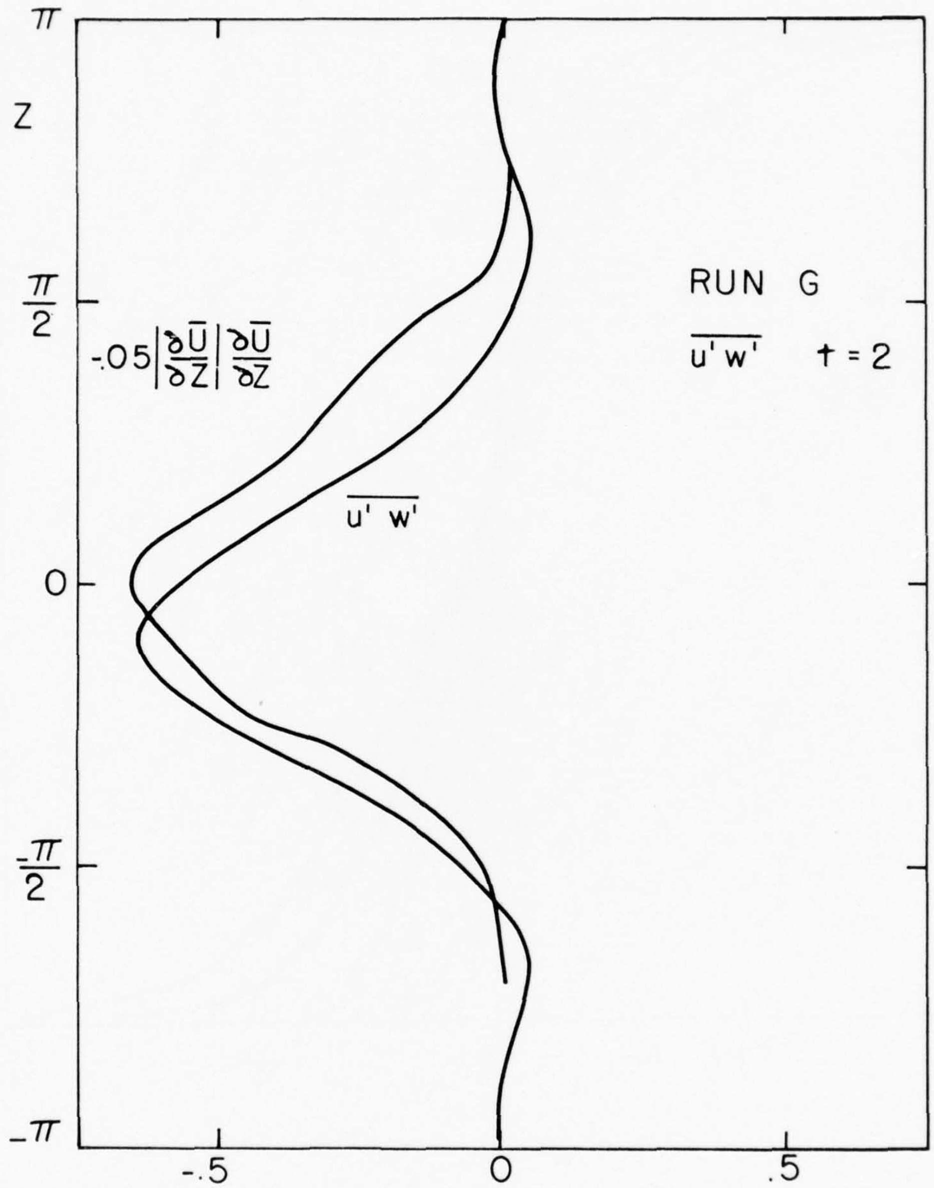

RUN G
$\overline{U}(z)$
t=2
t=1.3
t=.64
t=0
π
$\frac{\pi}{2}$
Z
0
$-\frac{\pi}{2}$
$-\pi$
-4
-2
0
2
4
$\overline{U}$

Figure 12.



π
Z
$\frac{\pi}{2}$
$-.05\left|\frac{\partial \bar{U}}{\partial Z}\right|\frac{\partial \bar{U}}{\partial Z}$
RUN G
$\overline{u'w'}$ t = 2
$\overline{u'\ w'}$
0
$-\frac{\pi}{2}$
$-\pi$
-.5
0
.5

Figure 13.

Some results of a simulation of a free turbulent shear layer are shown in Figures 12, 13. In this flow, all average quantities are functions of z alone, while the mean velocity $\overline{U}$ is in the x-direction; of course, fluctuating (turbulent) velocities exist in all three space directions. The mean velocity $\overline{U}$ is plotted as a function of z in Figure 12, for several values of the time t. Initially ($t = 0$), the mean velocity undergoes an abrupt jump near $z = 0$. The dashed curve would be the result for $\overline{U}(z)$ at $t = 2$ if (molecular) viscosity were totally responsible for the broadening of the shear layer. Instead, the actual rate of spreading is significantly faster, as shown by the results for $t = .64, 1.3, 2$, which is due to the effect of the turbulent velocities in diffusing the discontinuity.

This enhanced rate of diffusion of the turbulent flow over molecular diffusion is often characterized by an eddy viscosity coefficient. The validity of one particular type of eddy viscosity formulation, due to Prandtl (1925),[24] is tested by the results shown in Figure 13. Prandtl asserted that the Reynolds stress $\overline{uw}$ and the mean velocity $\overline{U}(z)$ are related by the mixing length hypothesis

$$\overline{uw} = -L^2 \frac{\partial \overline{U}}{\partial z} \left| \frac{\partial \overline{U}}{\partial z} \right| \qquad (17)$$

where L is the so-called mixing length, typically related to the large eddy size in the flow. The results plotted in Figure 13 show reasonably good agreement with the simple transfer expression (17) for the choice $L \approx .2$.

More sophisticated transfer models of turbulence may be similarly tested by numerical experiment. The results of these comparisons will be reported elsewhere.

COMPARISON WITH OTHER METHODS

There have been at least four parallel assaults on the turbulence problem which may be classified as:

1. analytical turbulence theories
2. turbulence transport models
3. sub-grid closure models
4. direct numerical simulations.

We have been discussing method (4) up to this point.

Method (1) includes all attempts at a fundamental theory of turbulence. The prototype examples are the direct-interaction appoximation (Kraichnan 1959)[8] and the test field model (Kraichnan 1971);[11] these theories are discussed critically by Orszag (1975).[18] They are characterized by their attempt to isolate and understand the fundamental difficulties of turbulance, i.e. the closure problem, nonlinear interaction, strong interaction, etc., usually without any free or adjustable parameters. These theories are nearly universally statistical in nature, the fundamental dynamical quantities being averages of the turbulent flow variables, like moments, average response functions, etc. These theories are very complicated and they have been applied only to homogeneous turbulence, one notable exception being the work of Herring (1969)[6] on large Prandtl number thermal convection. The extensions of these theories to general turbulent shear flows is quite difficult, although algorithms have been recently developed that should permit the economical application of these theories to a variety of shear flows.

Turbulence transport models are reviewed by Launder and Spaulding (1972)[13] and Harlow (1973).[5] These theories seek simplified models of turbulent flows, hopefully based on good physical insight, that are capable of application to complicated flows encountered in engineering

practice. Several adjustable parameters typically appear that must be chosen by comparison with experimental data. In common with the analytical theories, only averaged turbulent fields appear as dynamical quantities in these transport models. These smoothed fields exhibit symmetries (like axisymmetry for a circular jet flow in unstratified fluid) that the detailed, unaveraged, turbulent flow fields to not themselves exhibit. Thus, instead of having to perform a three-dimensional time dependent simulation, it may be possible to reduce the problem to a steady state problem in fewer than three space dimensions with consequent enormous savings of computational effort, (assuming numerical solution of the equations of the theory is ultimately necessary).

In contrast to the analytical theories, the transport models deal directly with only the gross properties of the turbulent flow, with little or no attempt at consideration of the fundamental dynamics of interaction between the various turbulent scales of motion. The advantage of this approach over the analytical theories is clear: even for complicated inhomogeneous turbulent flows, the dynamical quantities appearing in the transport models are relatively simple. The disadvantage is equally clear: potentially important dynamical information is forever thrown away by the cavalier disregard of detailed dynamics.

There is a very wide variety of transport models now being touted in the literature. All authors have in common the use of the Reynolds equation for the mean velocity field obtained by averaging the Navier-Stokes equations. This equation relates the evolution of the mean velocity to the Reynolds stresses. Second-order closure models develop further equations of motion for the Reynolds stresses themselves. However, these additional equations do not contain enough information to predict evolution of the turbulence without additional hypotheses, just because this further evolution depends in an essential way on the details of the flow that were wiped out by averaging. In the end, several ad hoc

assumptions relating dynamical quantities must be made to close the equations of these models; all differences between these models enter from the precise closure conditions that are chosen.

In the subgrid closure models (method 3), turbulence transport approximations are made only on those scales of motion not explicitly resolved by a numerical approximation to the Navier-Stokes equations. The chief architect of these methods has been Deardorff (e.g. 1970a, b),[2,3] who has applied both simplified eddy viscosity closures and sophisticated second-order transport closures to the subgrid component of the flow. The large scales (those explicitly resolved by the numerical approximation) are treated as they are in direct numerical solution of the Navier-Stokes equations. The very small scales are treated by statistical approximation, while the large scales are treated in detail. The effect of unresolved small scales on larger resolved scales is represented by an eddy viscosity K replacing molecular viscosity. Here K is chosen (Smagorinsky et al 1965)[26] as

$$K = (c\Delta x)^2 \left| \left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right)^2 \right|^{1/2}$$

in three dimensions, and

$$K = (c'\Delta x)^3 \left| \frac{\partial}{\partial x_i} (\nabla \times \vec{v}) \right|$$

in two dimensions; Δx is the spatial resolution and c, c' are constants (~.1-.2). These expressions for K are appropriate only if first- or second-order space differencing is used; higher-order schemes require different eddy viscosities.

This approach has very much to commend it, including the important fact that there is no direct Reynolds Number limitation on the simulations, since the effect of the subgrid turbulence is taken into account.

It is the author's opinion that the subgrid approach must eventually be involved in solution of all turbulence problems, since one is interested in details of the large scales and not just their statistical properties. However, the current methods for inclusion of the subgrid effect have a number of disadvantages, including the use of ad hoc turbulence transport models for the subgrid component and the neglect of any stochastic effect of fluctuations in subgrid flow variables on the generation of fluctuations in large scales.

It appears that the principal advantage of the subgrid approach over direct simulation is the absence of any Reynolds number limitations in the former. However, this is not so. The accuracy of the subgrid closures requires that the separation of the flow into subgrid scale and large scale components does not have a significant effect on the evolution of large scales.

In the same way, consider the possibility that direct numerical simulation (method 4) of a huge Reynolds number flow may be accomplished by artificially decreasing the Reynolds number to the point where the flow can be accurately simulated on existing machines. Of course, it is not possible that all scales of motion of the reduced Reynolds number flow be unchanged, but it is possible that sufficiently large scales be unchanged by the Reynolds number change. In fact, large scale features of turbulent flows do not seem to change with Reynolds number, if boundary and initial conditions are fixed independent of Reynolds number (Herring et al 1974).[7] Some evidence for this behavior is shown in Figure 14. In Figure 14, we plot the evolved vorticity contours of the initial vorticity field described in the caption to Figure 5 for three different Reynolds numbers. The initial flow field is identical for all three Reynolds numbers. The similarity in large scale structure is apparent.

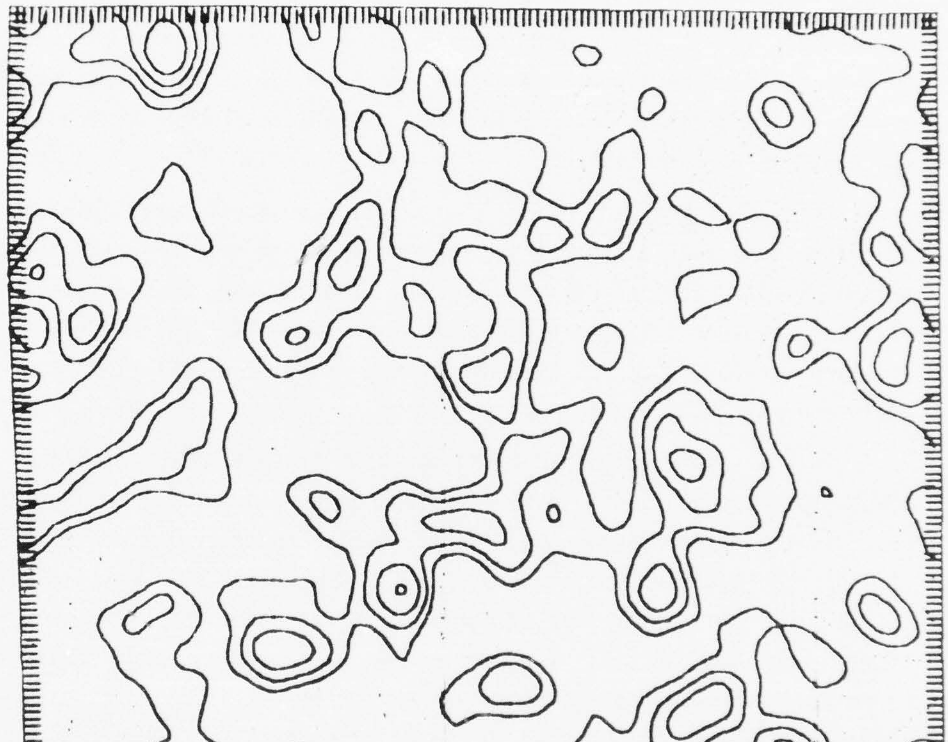


Figure 14a.

Figure 14b.

Figure 14c.

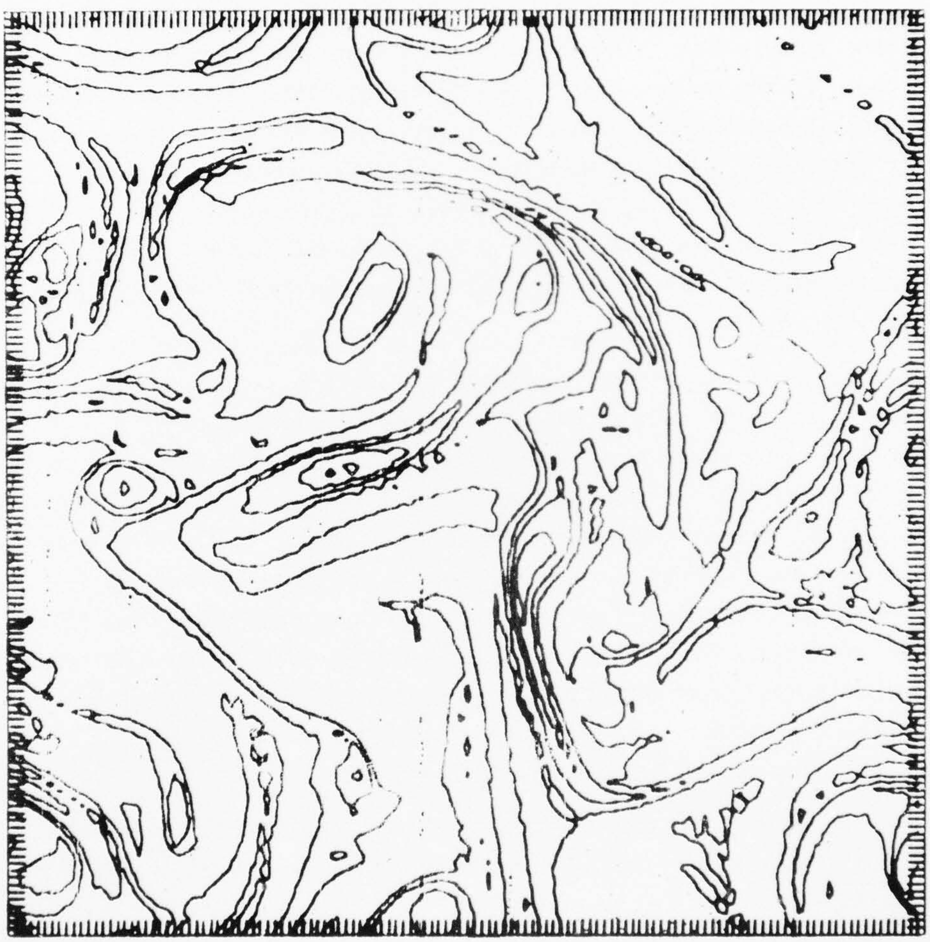


Figure 14d.

As a consequence, it does not seem necessary to simulate huge Reynolds-number turbulence to gain information on large and moderate scales of motion. It is only necessary to simulate flows at Reynolds numbers at which the desired scales of motion have achieved Reynolds number independence. This behavior is illustrated pictorially in Figure 15. As the Reynolds number increases, the small scale of three-dimensional turbulence adjusts to maintain the energy dissipation rate (cf. PROBLEMS), giving a mean-square vorticity spectrum $k^2E(k)$ that extends to higher and higher k. The peak in $k^2E(k)$ occurs near $k \sim 1/\ell$, where ℓ is given by (10). For $k \lesssim 1/\ell$, the flow is nearly Reynolds number independent, not just statistically but apparently in detail as well.

Some results of Deardorff's (1970b)[3] simulation of the planetary *boundary* layer are plotted in Figure 16. Here K_m is the coefficient of subgrid scale eddy viscosity (suitably normalized); $K_{MX} = K_m - \overline{uw}/\partial\bar{u}/\partial z$ and $K_{MY} = K_m - \overline{vw}/\partial\bar{v}/\partial z$ measure the effective eddy viscosity due to the Reynolds stresses of resolved small scales acting on resolved large scales; K_θ is a similar measure of the eddy conductivity of heat due to resolved scale motions. It is apparent from Figure 16 that the Reynolds stresses due to resolved motions are much larger than those due to the imposed subgrid eddy viscosity. Consequently, most of the eddy transport is being accomplished by the resolved scales with the subgrid component providing only the necessary small-scale excitation to ensure the Reynolds number independence of energy dissipation.

In summary, it appears that the large scales of turbulent flows have a strong tendency to adjust themselves to be independent of the details of the dissipation mechanism; subgrid eddy motions or molecular viscous action are both effective dissipators provided that the viscosity is small enough that the large scales have achieved Reynolds number independence. It seems that either method (3) or (4) can be used to achieve simulations of turbulent flows with limited resolution; the

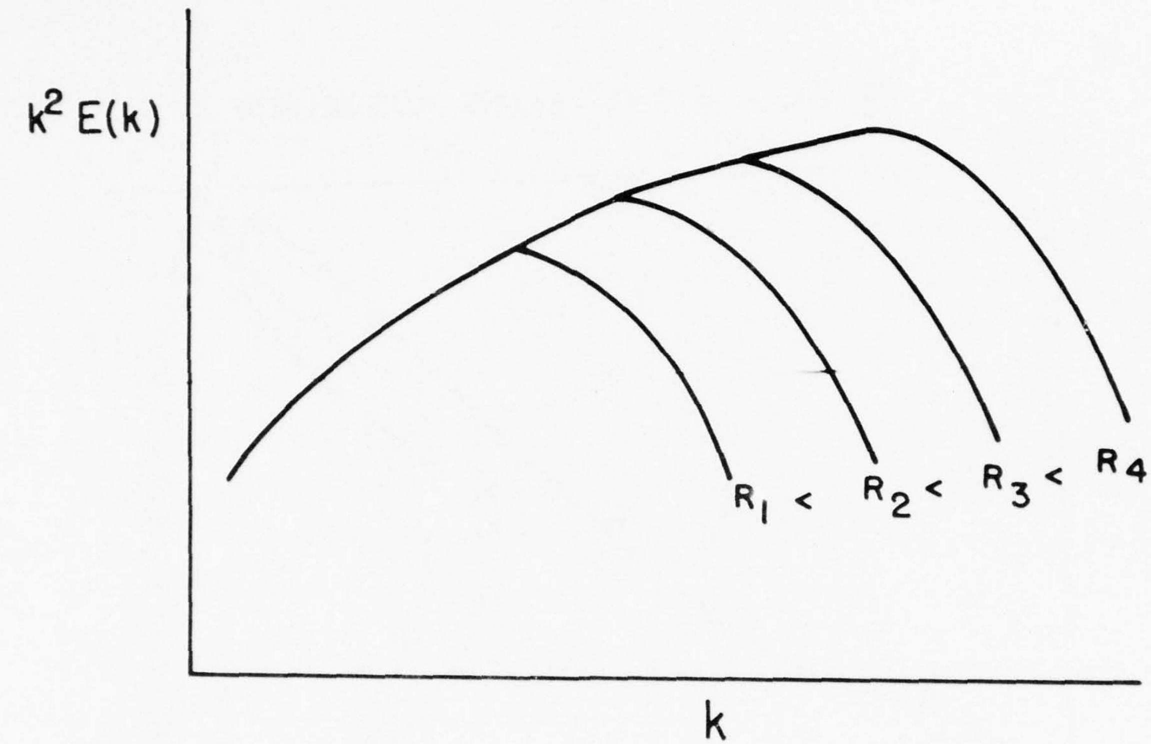
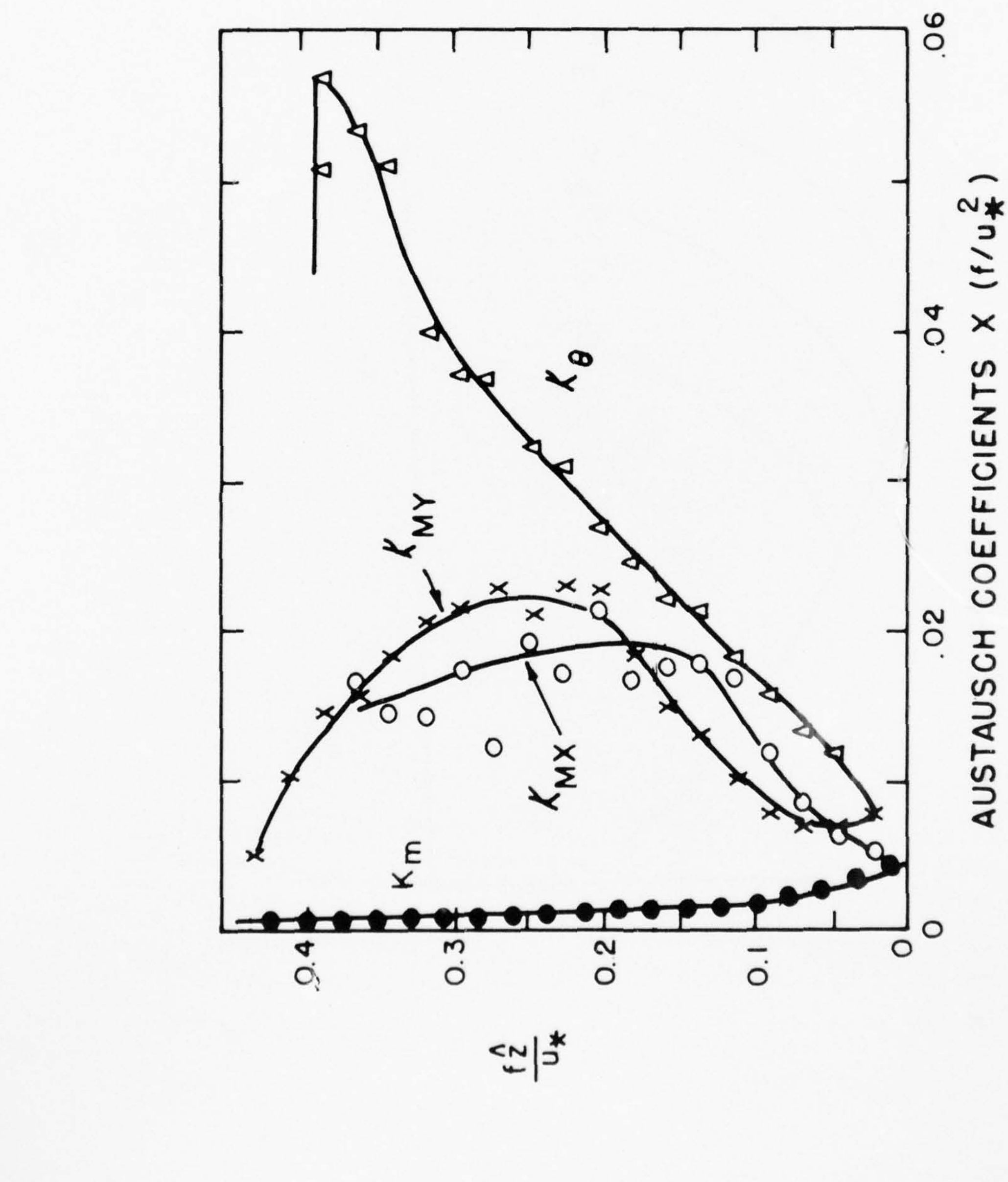


Figure 15.



$\frac{f\hat{z}}{u_*}$
0
0.1
0.2
0.3
0.4
.02
.04
.06
AUSTAUSCH COEFFICIENTS X (f/u_*^2)
K_m
K_{MX}
K_{MY}
K_θ

Figure 16.

outstanding question concerns the efficiency with which invariance to the scale of separation into subgrid and resolved motions (for method 3) or Reynolds number independence (for method 4) is achieved.

PROSPECTS

Computers that will be available by 1980 are not likely to be more than an order of magnitude or so faster than those available today. Reynolds number restrictions similar to those imposed on current simulations will still be present. However, the order of magnitude improvement in speed can be used to give an order of magnitude improvement in statistics, a prospect which is especially important for inhomogeneous turbulence simulations.

It seems that the best hope for achieving simulations of huge Reynolds flows is to investigate further effects like Reynolds number independence. These effects should be used to give both moderate Reynolds number simulation models and subgrid closure models having the property that they give essentially the same evolution for suitably large scale eddies as the required huge Reynolds number flow.

Techniques must also be investigated for simulation of flows in the presence of solid boundaries when the available resolution is not sufficient for an exceedingly thin turbulent boundary layer. The latter problem has not received as much attention as the (interior) turbulence modelling problem, through it is equally important. One possible approach is the consideration of small sections of the flow in detail, then parametrizing their properties and finally the use of the parametrization to provide boundary conditions in the large scale simulation (as done by Deardorff 1970b in his parametrization of the planetary boundary layer for use in large-scale general circulation models of the atmosphere).[3]

In conclusion, there has been some modest progress towards the solution of the problem of numerical simulation of turbulence. However, many interesting and important problems remain to be solved, hopefully in the not too distant future.

REFERENCES

1. Batchelor, G.K., (1953), "The Theory of Homogeneous Turbulence," Cambridge University Press.
2. Deardorff, J.W., (1970a), "J. Fluid Mech.," 41, 363.
3. Deardorff, J.W., (1970b), "Geophys. Fluid Dyn.," 1, 377.
4. Grant, H.L., R.W. Stewart and A. Moilliet, (1962), "J. Fluid Mech.," 12, 241.
5. Harlow, F.H., (1973), "Turbulence Transport Modelling," Amer. Inst. Aero. Astro., New York.
6. Herring, J.R., (1969), "Phys. Fluids," 12, 39.
7. Hering, J.R., S.A. Orszag, R.H. Kraichnan and D.G. Fox, (1974), "J. Fluid Mech.," 66, xxx.
8. Kraichnan, R.H., (1959), "J. Fluid Mech.," 5, 497.
9. Kraichnan, R.H., (1966), "Phys. Fluids," 9, 1728.
10. Kraichnan, R.H., (1967), "Phys. Fluids," 10, 1417.
11. Kraichnan, R.H., (1971), "J. Fluid Mech.," 47, 513.
12. Kriess, H.O. and J. Oliger, (1973), "Methods for the Approximate Solution of Time Dependent Problems," GARP Publication Serial Number 10, World Meteor. Org.
13. Launder, G. and D.B. Spalding, (1972), "Mathematical Models of Turbulence," Academic Press.
14. Leith, C.E., (1971), "J. Atmos. Sci.," 28, 145.
15. Orszag, S.A., (1971), "Stud. Appl. Math.," 50, 395.

16. Orszag, S.A., (1974a), "Mon. Weath. Rev.," 102, 56.

17. Orszag, S.A., (1974b), in "Computer Methods in Applied Sciences and Engineering," Part 2, Springer-Verlag, p. 50.

18. Orszag, S.A. (1975), "Lectures on the Statistical Theory of Turbulence," Les Houches Summer School, Springer-Verlag.

19. Orszag, S.A. and M. Israeli, (1974), "Ann. Ref. Fluid Mech.," 6, 281.

20. Orszag, S.A., M. Israeli and L. Jayne, (1975), to be published.

21. Orszag, S.A. and Y.-H. Pao, (1974), "Adv. in Geophys.," xxx.

22. Orszag, S.A. and G.S. Patterson, (1972), "Phys. Rev. Letters," 28, 76.

23. Patterson, G.S. and S.A. Orszag, (1973), "Atmos. Tech.," 3, 71.

24. Prandtl, L., (1925), "Zeit. f. angew. Math. u. Mech.," 5, 137.

25. Saffman, P.G., (1971), "Stud. Appl. Math.," 50, 377.

26. Smagorinsky, J., S. Manabe and J.L. Holloway, (1965), "Mon. Weath. Rev.," 93, 727.

SESSION III

COMPUTER CAPABILITY

Preceding Page BLANK

SOME ASPECTS OF COMPUTING WITH
ARRAY PROCESSORS

C. E. Leith
National Center for Atmospheric Research
Boulder, Colorado

INTRODUCTION

An array processing computer represents a major change in hardware organization from the conventional serial computers with which we are familiar. Many of the numerical methods and programming techniques that have been developed along with the development of serial machines must therefore be reconsidered in the light of this change, and, indeed, such a reevaluation of old methods and development of new methods is going on at many computing and scientific research centers.

It is the purpose of this paper to describe in general terms the ways in which array processing computers differ from serial computers, the possibilities for increased computing speed arising from such hardware changes, and the kinds of computer programs that will be able to achieve the potential speed of these new machines. This is intended to be only a user's introduction to array processors; for specific technical details one must turn to the rapidly expanding literature on the subject.

NUMERICAL MODELS

A principal demand for greater computing speeds is for the kind of calculation involved in the simulation of geophysical fluid flow by means of a numerical model. Common examples are atmospheric and oceanic general circulation models, atmospheric boundary layer models, and models of mountain waves and clouds. Although many different numerical techniques

have been used to construct such models, they all have certain common characteristics of importance to computer design.

In a numerical model the state of the physical system is specified at a given model time by the numerical values of a large but finite number, P, of prognostic variables. The number P is a fundamental measure of the size of a model; it is the number of values that must be specified in order to start a calculation or that must be saved to interrupt and restart a calculation. We may think of P as the dimensionality of the model's phase space; the state of the model is represented at any time by a P-component position vector.

The evolution of time, t, in the physical system is modeled by a sequence of calculational time cycles. In a single time cycle each of the P prognostic variables is advanced by numerical process to a value appropriate to a time that is later by a time step Δt. The execution of a time cycle will require a large number, N_a, of arithmetic operations, and the number N_a is a measure of the computing power required. The ratio $n_a = N_a/P$ is a measure of the arithmetic complexity of a model. It can be on the order of ten for highly simplified models but is typically on the order of 100 for more realistic general circulation models. The parameter n_a is important in deciding on the proper balance between the speed of execution of arithmetic operations and the speed of access to the memory holding the P prognostic variables.

SERIAL COMPUTERS

A serial computer consists (in a highly simplified description) of three components, as shown in Figure 1. The memory (M) holds values of prognostic and working variables as floating-point numbers, and also holds the set of instructions that defines the numerical model. The arithmetic unit (AU) carries out arithmetic operations on operands supplied from the memory and returns results to the memory; the two-way

AD-A043 448   NATIONAL ACADEMY OF SCIENCES-NATIONAL RESEARCH COUNCI--ETC   F/G 20/4

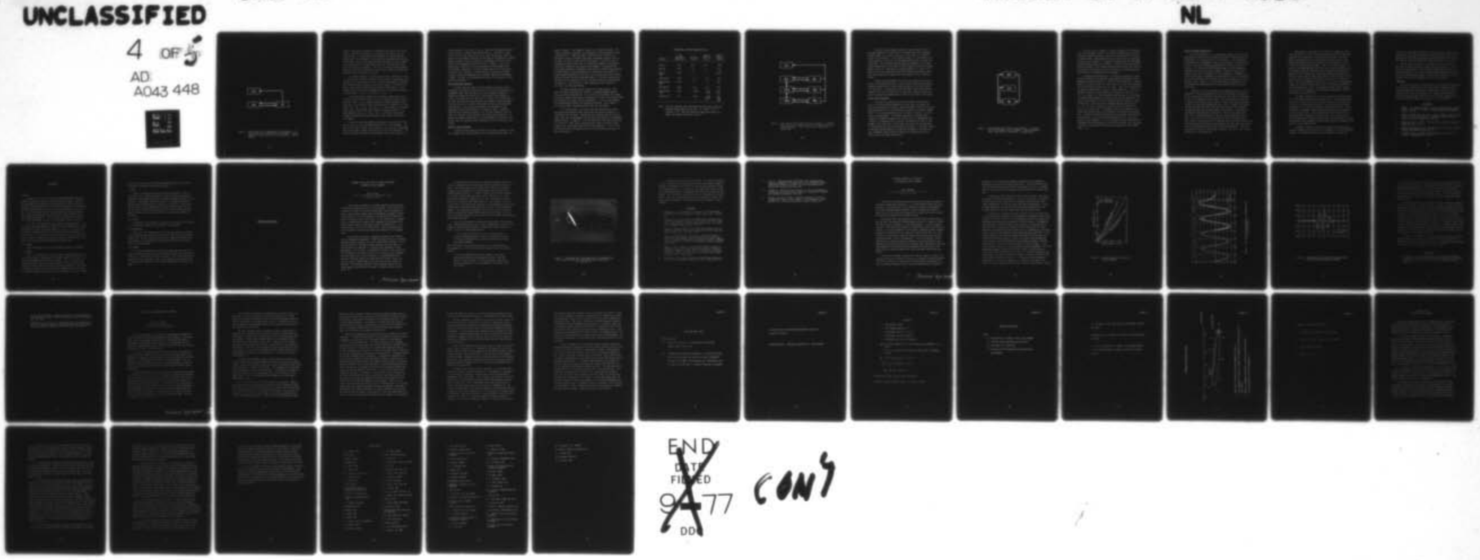
PROCEEDINGS OF THE WORKSHOP ON NUMERICAL HYDRODYNAMICS.(U)

JUL 75   N00014-67-A-0244-0013

UNCLASSIFIED   NL

4 OF 5

AD A043 448



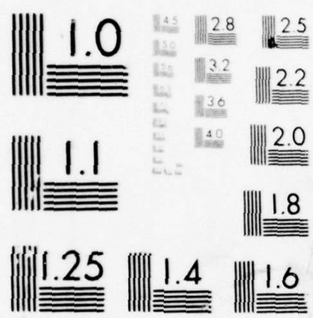
END

DATE FILMED

9-77

DDC

CON'T



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

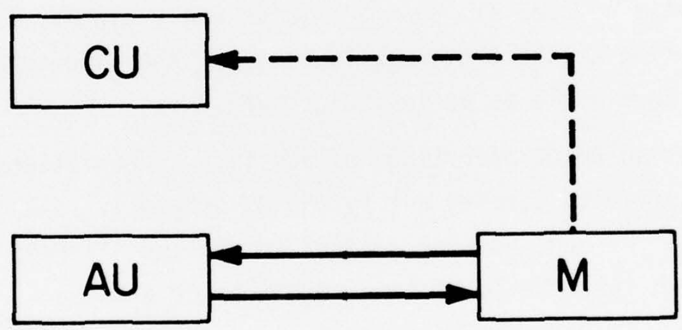


Figure 1. Block Diagram for a Conventional Serial Computer. CU, Control Unit; AU, Arithmetic Unit; M, Memory. Flow of Operands -- Solid Lines; Flow of Instructions -- Dashed Lines.

flow of floating-point numbers is indicated by the solid lines in the figure. Control of this process is vested in the control unit (CU) which fetches its instructions from the memory as indicated by the dotted line. The CU decodes an instruction to determine which operands are involved, fetches the proper operands from memory to the AU, determines the operation to be performed by the AU, and stores the result back from the AU to the memory. The CU is also able to choose between alternate algorithms, depending on the outcome of numerical comparisons which have been carried out in the AU.

The computer memory is usually divided into a hierarchy of levels of increasing word capacity and word access time but of decreasing cost per word. For our purposes the level in which the prognostic variables are stored must have a capacity of at least P words. In order to complete a time cycle we need at least P fetches and P stores. If the average access time at the P-storage level is τ_m then the total memory-accessing time in a cycle is at least $T_m = 2P\tau_m$.

The floating-point operations of addition, subtraction, multiplication, and division carried out by the AU generally take different execution times. We can, however, define an average arithmetic operation time τ_a such that the total time required for arithmetic in a time cycle is $T_a = N_a\tau_a = Pn_a\tau_a$. The time τ_a is usually about five hardware clock cycles. For optimal use of computer hardware we want $T_a = T_m$ which requires that $\tau_a/\tau_m = 2P/N_a = 2/n_a$. For models with greater arithmetic complexity we need relatively slower access time to the P-level memory.

During the past two decades many computers have been built that more or less fit our simplified description of a serial computer. The table lists eight of these, each of which, in its time, has represented one of the fastest computers available for scientific computation. The listed fixed-point addition times and memory-level access times give

rough estimates of the times τ_a and τ_m. Both have been getting dramatically shorter, τ_a perhaps more rapidly than τ_m. But there is some possibility that this historical rate of speed increase must now slow down since the inherent limitation on the speed of propagation of electromagnetic signals, namely, 30 cm/nsec, is beginning to be felt in computers with hardward cycle times of 30 nsec. However, a way of bypassing this limitation is to use concurrency in both memory access and arithmetic operations. This has already been done to some extent in recent serial computers in which efforts are made to have memory accesses, instruction decoding, and arithmetic operations overlapping in time. The idea is being pushed much further in the design of array processing computers.

ARRAY PROCESSING COMPUTERS

In many numerical models the calculational time cycle can be decomposed into subcycles, each of which in turn can be decomposed into independent tasks that are identical in the arithmetic sequence or algorithm followed and differ only in the values of the operands involved. We assume here that the tasks in a subcycle are independent in the sense that they may be performed in any sequence without changing the final result; it is obvious then that the tasks can be performed simultaneously. Array processing computers are designed to take advantage of any such parallelism in a numerical model. Presently designed array processing computers fall into two categories which we shall call parallel array processors and string array processors. Although the hardware organization is quite different for these two categories, they are logically nearly equivalent so far as the user is concerned.

PARALLEL ARRAY PROCESSORS

A parallel array processor differs from a serial computer in that the arithmetic unit (AU) and memory (M) are replicated many times, as

shown in Figure 2. The number of copies, W, is called the width. The primary example of a parallel array processor is the one-quadrant Illiac IV (Barnes et al., 1968;[1] McIntyre, 1970[5]) which has a width W = 64. Each AU carries out arithmetic operations on operands supplied from its own memory to which it then returns the results. A single CU acts on a sequence of instructions that is distributed through all the memories. After the CU fetches an instruction from a memory, it determines the local memory addresses of operands, simultaneously fetches proper operands from each memory to the corresponding AU, determines the operation to be performed simultaneously by all AUs, and simultaneously stores results from each AU back to the corresponding memory. On a given calculational step each AU must generally carry out the same arithmetic operation that all others do. The exception is that, depending on the results of a local test, an AU may do nothing.

At times an AU may need an operand from another memory. This need is satisfied indirectly by transfer of operands between memories using a process called routing. In a routing cycle operands are fetched to the AUs, moved to AUs that are neighboring in a one- or two-dimensionally cyclic sense, and stored in their new memories. In this way W operands are simultaneously shifted into neighboring memories. By a number of such routing steps any operand can be moved into any memory.

It is obvious that if all AUs can be kept usefully busy, then a calculation can proceed W times faster than one using the same AU in a conventional serial design. Keeping AUs busy requires that the calculational time cycle be decomposed into subcycles consisting of W independent identical tasks. In general, if we can subdivide a time cycle consisting of a total of r tasks into s' subcycles each of which contains at most W tasks, then the average number of tasks per subcycle w' = r/s' is the parallelism of the calculation and w'/W is the efficiency with which it can be carried out.

CONVENTIONAL COMPUTER CHARACTERISTICS

Computer	First Delivered (Month/Year)	Add Time (μsec)	Memory Cycle Time (μsec)	Memory Capacity (Words)
Univac 1	3/51	282	242	1,000
IBM 704	12/55	24	12	32K
Univac LARC	5/60	4	4	30,000
IBM Stretch	5/61	1.5	2.2	96K
Control Data 6600	9/64	0.3	1.0	128K
IBM 360/91	2/67	0.18	0.75	512K
Control Data 7600	1/69	0.0275	0.275 1.760	64K 512K
IBM 360/195	2/71	0.054	0.054 0.810 8	32K 256K 16M

Note: In recent computers some concurrency has been used to decrease both add time and memory cycle time. $K = 1{,}024 = 2^{10}$, $M = 1{,}048{,}576 = 2^{20}$. Data from Charles W. Adams Associates, Inc. (1967)[2] and Keydata Corporation (1971).[4]

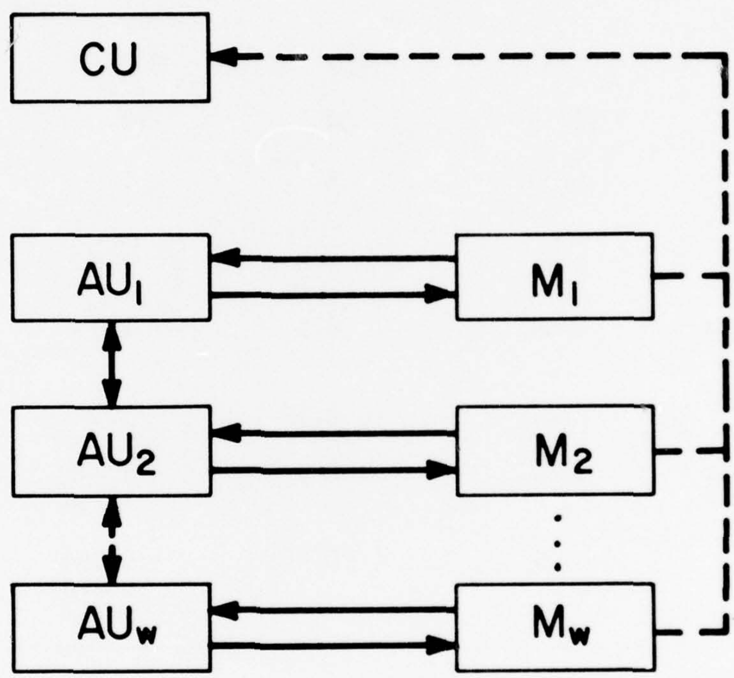


Figure 2. Block Diagram for a Parallel Array Processor. CU, Control Unit; AU_1, i-th Arithmetic Unit; M_1, i-th Memory; W, Width. Flow of Operands -- Solid Lines; Flow of Instructions -- Dashed Line.

A parallel array processor provides a considerable saving in cost when compared to a collection of W conventional serial computers. Although there is no saving in the cost of AUs, there is an obvious saving in the shared CU. A less obvious saving is in the smaller individual memories. The Illiac IV individual memories have a capacity of 2,048 words. This capacity would be woefully inadequate in a conventional computer but is sufficient for local memory and provides a total memory of 64 x 2K = 128K words (in binary computers $K = 1,024 = 2^{10}$). There is also a considerable saving from sharing the peripheral equipment that accounts for a major part of the total cost of a computing system.

The most severe constraint on the use of a parallel array processor is the requirement that a single CU must control many processors. Hardware configurations in which each processor has its own CU avoid this constraint. Such designs are called simply "parallel processing computers" and are not actually array processors; they have their own special programming problems which we shall not consider in this paper.

STRING ARRAY PROCESSORS

A string array processor, such as the STAR or the ASC being built by the Control Data Corporation and the Texas Instruments Corporation, respectively, achieves a speed comparable to that of a parallel array processor but in a quite different way (see, for example, Graham, 1970).[3] A string array processor has, as shown in Figure 3, a single AU and a single memory. An arithmetic operation is divided into a sequence of elementary stages, each carried out in an independent part of the AU. An operation with a new pair of operands can start in each hardware clock cycle, but it takes many such cycles for the operation to be completed and the result to become available. To provide a stream of operands rapidly enough to feed the AU, the memory is divided into many independent banks from which operands are fetched and back to which results are stored sequentially.

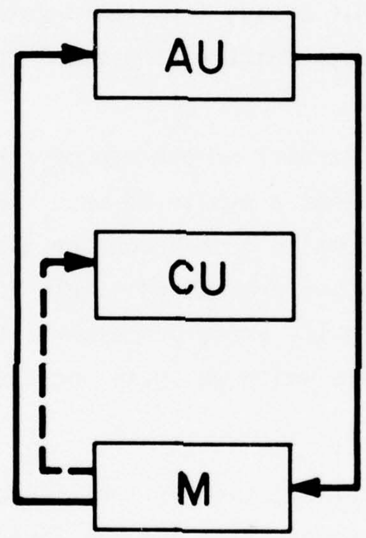


Figure 3. Block Diagram for a String Array Processor. CU, Control Unit; AU, Arithmetic Unit; M, Memory. Flow of Operands -- Solid Lines; Flow of Instructions -- Dashed Lines.

As with a serial computer, a separate sequence of instructions flows from the memory to the CU, but now an instruction may initiate a vector arithmetic operation in which the same operation is carried out repetitively, using operands from two linear arrays and producing a linear array of results. An imaginary pipeline runs from the memory through the AU and back to the memory. The time to feed new operands into the pipeline is much shorter than the total transit time, and, if the pipeline can be kept full, the calculation can proceed many times faster than in a conventional serial computer.

Although the organization of a string array processor is quite different from that of a parallel array processor, both make similar demands on program parallelism. Since the vector operation in general cannot use an early result in a later step, it is logically equivalent to the simultaneous execution of all steps. There is no limitation on the number of operations or steps in a vector operation. The time τ_n needed to complete a vector operation of length n is given by $\tau_n = (n_o + n)\tau_a$, where τ_a is the individual step time and n_o is the number of overhead steps required to start and stop a vector operation. Consider a calculational time cycle characterized by a total of r tasks divided into s subcycles with a consequent program parallelism of $w = r/s$. For such a program the efficiency of task execution is $r/(n_o s + r) = w/(n_o + w)$. The number n_o serves as a measure of computer parallelism similar to the number W for parallel array processors. In order to achieve an efficiency of use greater than one-half, we must have $w > n_o$ in a string array processor and $w' > W/2$ in a parallel array processor. It is to be hoped that some overlapping of overhead cycles with execution cycles can be achieved, but in any case efficiency of execution will be seriously degraded for calculations with $w < n_o$.

PARALLEL PROGRAM ORGANIZATION

With array processors it is evident that there can be a serious degradation of efficiency for programs that have not been specifically organized to fit the parallelism of the computer. It will therefore be important for a programmer to know how his program is being executed and not to have this knowledge concealed behind the intricacies of a compiler. For this reason the FORTRAN language is being extended by the addition of explicit vector arithmetic statements which will be compiled into vector instructions on array processors. FORTRAN programs to which vector statements have been added will still be suitable for serial computers since for these the vector statements will be compiled as loops. There is, in fact, some increase in efficiency with the Control Data 7600 machine arising from the more effective use of stack loops.

A general rule for the conversion of programs written for serial computers is that those programs will be most efficiently transferred to an array processor that have long parallel inner loops in which most of the serial computing time is spent. For such programs the transfer is made by replacing the inner loop by a collection of vector operations in which the loop index becomes the vector index. Evidently the longer the loop the more efficiently the vector operation will be executed. An important constraint on these inner loops, however, is that they be parallel and not sequential or inductive. That is, the loop must be such that it could be executed in any order without modifying the result. This rules out, as an inner loop, the sort of inductive loop that arises in the inversion of tri-diagonal matrices by back-substitution; each step requires the result of the previous step, but these most recent results are generally not available from vector operations. The exception would be a special inner product operation, but this is not directly applicable to the present case.

Many models for the numerical simulation of geophysical fluid flows are based on a finite difference mesh with an explicit marching scheme for the integration of the equations of motion. For these models the general rule will require that the inner loop be in the longest mesh direction and that comparison tests leading to choices of alternate algorithms be avoided. Thus it will be desirable, if possible, to impose boundary conditions not as special algorithms but as the general interior algorithm; special operand values should be used even if they require the addition of a few exterior mesh points. The only logical decision that can be made with array processors is to execute or not to execute the operation called for at a particular step. The decision depends on the value (1 or 0) of the corresponding element in a logical control vector, and it is obviously inefficient to skip exectuion at a step. In atmospheric models special thought must be given, for example, to the treatment of mountains, convective adjustment, and precipitation.

For models based partly on implicit numerical algorithms there will be the special problems mentioned earlier. For multidimensional models such implicit relations are often confined to one dimension at a time, as in alternating-direction or fractional-time-step schemes. The general rule is that the innermost loop must be explicit; that is, each step in an implicit sweep should be executed for all elements in an explicit inner loop. Such an organization of the calculation requires additional storage for working vectors and, in the case of alternating-direction methods, requires some careful planning to effect the required array transpositions efficiently, which will depend on the specific array processor used.

A number of people are working on spectral or pseudo-spectral transform methods as a more efficient replacement for finite difference methods. Such methods benefit from the regularity of domain and

uniformity of algorithm desirable for array processing. Although array transposition may cause some problems, there is general optimism that the necessary fast-Fourier-transform algorithms can be worked out, and that spectral transform methods will use array processors efficiently.

Certain other new numerical algorithms seem to be evolving in ways that are not particularly suitable for array processors. For example, the development of finite-element methods based on a mesh of irregular topology or of various semi-Langrangian hydrodynamic schemes with fluctuating nearest-neighbor relations would seem to pose serious problems for effective implementation on an array processor.

CONCLUSION

The efficient use of array processors will provide new constraints on the choice of algorithms for the construction of numerical models. Some algorithms will be completely inappropriate, and some, such as those involving implicit techniques, will require careful planning. But large numerical models based on explicit marching schemes will be readily converted to make efficient use of an array processor.

REFERENCES

1. Barnes, G., R. Brown, M. Kato, D. Kuck, D. Slotnick and R. Stokes, (1968), "The Illiac IV Computer," IEEE Transactions on Computers C-17(8), 746-757.

2. Charles W. Adams Associates, Inc., (1967), Computer Characteristics Table in "Digital Computer User's Handbook," Melvin Klerer and Granino Korn, Eds., McGraw-Hill, New York.

3. Graham, William R., (1970), "The Parallel and Pipeline Computers," Datamation 16(4), 68-71.

4. Keydata Corporation, (1971), "Characteristics of Digital Computers," Computers and Automation 20 (6B), 24.

5. McIntyre, David E., (1970), "An Introduction to the Illiac IV Computer," Datamation 16(4), 60-67.

DISCUSSION

J. Boris

I would like to correct a possible misinterpretation which came through in your talk, but one I am sure you didn't intend. Some of us may have gotten the impression that the numbers on the Star are more firmed up than those on the ASC. The fact of the matter is that the ASC has been running programs for over a year whereas Star has yet to run one. So, the squiggly brackets on your draft don't mean that they're not running, only that they are adjusting the speed of the machine as they improve the hardware. The Star, which is supposed to be holding for a fixed performance, is not, to my knowledge, actually running.

Also, the FFT problem that you mentioned is being worked on. The speeds that have been obtained in the case of the benchmark FTT's are on the order of 20 times faster than 360/91 for an ASC. If the Star runs according to specifications it will handle the same problem about 30 times faster. But reaction time in these machines is cricual and, unless we are careful, running time of some solutions can be dominated by such factors as start-up time as mentioned by Professor Leith.

S. I. Cheng

Would you expand a bit on your reference to memory requirements?

C. E. Leith

Well, as I mentioned, the level of memory required depends on the size of the problem. When you get up around a quarter of a million you are getting beyond the local fast core memory. Yet, in the 7600, for example, there are relatively rapid transfers if you do things in the sequential fashion, and if you organize the work properly. I guess what I am saying is that you are not IO bound in between the small core and the large core memory on the 7600 if you are careful, but we will always

have to keep in mind exactly these same problems when we go to faster arithmetic with parallel processing computers.

J. Boris

There is another point worth mentioning about memory: That is, the memory size of these new machines is going to have to be greater relative to their speed ratio than is generally thought, based on current machines. For example, vectorizing implicit tri-diagonal solutions employs binary folding which will require a large array. Two or three scratch arrays will be needed which presently we don't require at all. So, the price of getting the high speed will be two to three times the memory, and that is not to mention what you will want to transfer off the discs.

C. E. Leith

Incidentally, this machine is planned for one to four million words in a 48 nanosecond memory in anticipation of that problem.

R. W. MacCormack

Because of the architecture of these machines Fortrand or general purpose languages may not be desirable in that they lead to inefficient machine use. At Ames, when we got the Illiac computer we developed our own language called Computational Fluid Dynamics (CFD) and realized a factor of two improvement.

C. E. Leith

As I mentioned before, once you have decided that you have to re-organize things appropriate to array processing anyway, then you realize that, say, in the case of the 7600, you might as well affect these vector operations in the stack loop. And, as you suggest, routines worked out at Livermore have yielded a factor of two improvement in the use of the 7600.

INFORMAL CONTRIBUTIONS

A COMMENT ON THE EFFICIENCY OF FINITE-DIFFERENCE METHODS IN FLUID DYNAMICS

Hans J. Lugt
Naval Ship Research and Development Center
Bethesda, Maryland

I would like to comment on a problem regarding the efficiency of numerical methods being used in fluid dynamics. There seems to be no universal recipe for solving the equations of motion numerically. Even if such a general method could be developed, techniques tailored to each specific class of problems can be much more efficient (with regard to computer time) and accurate. In this connection, it appears to me, a communication problem arises regarding the dissemination of new or improved numerical techniques among fluid dynamicists. To illustrate this, I wish to call attention to a class of methods which are very powerful but which have been much neglected by fluid dynamicists.

For more than five years a developing program in the Computation and Mathematics Department of NSRDC has been under way to generate and apply computer programs for solving hydrodynamic problems. In this effort solutions of the Navier-Stokes equations also have been constructed which are important for studying local phenomena in ship hydrodynamics (for instance, to determine the onset of flow instability, the flow behavior near and in suction slots, and the vortex shedding from protruding parts of naval vessels). By experimenting with a number of different methods, we found that fast Poisson solvers are very efficient, although they are restricted to certain classes of problems. These methods appeared in the literature during the last decade and are connected with the names of Buneman, Hockney, Golub, et al.[1,2,3]

PRECEDING PAGE BLANK

To demonstrate the efficiency of these fast methods and the complexity of problems which now can be solved, the following example is given: the laminar flow of an incompressible fluid past an abruptly started plate translating with constant speed U while rotating with constant angular velocity Ω. Figure 1 displays the streamline patterns at a certain instant after the start for Re = 200 and Ro = 2, where the Reynolds number Re is defined by $Re = dU/\nu$ and the Rossby number Ro by $Ro = 2U/\Omega d$ with d the plate width and ν the kinematic viscocity of the fluid. The solution was obtained for the streamfunction-vorticity formulation of the Navier-Stokes equations in which a Poisson equation appears. The comparison of Hockney's fast Poisson solver with the successive overrelaxation method (SOR) showed that, with equal accuracy, this flow problem could be solved more than 50 times faster with Hockney's method than with SOR![4,5]

Such fast Poisson solvers have barely been used by fluid dynamicists. Noteworthy exceptions are a study of compressible inviscid flows by Martin and Lomax[6] and a solution of the Navier-Stokes equations by Otte.[7] In fact, in 1970 Buneman wrote:[8]

> "...This involves solving the elliptic Poisson equation, a time consuming operation for which fluid dynamicists have, in the past, employed what to plasma simulators seem clumsy and outdated methods."

This forthright criticism did not seem to have a visible effect on fluid dynamicists. For in 1974 Orszag and Israeli wrote in a review paper:[9]

> "It is now recommended that direct methods, rather than relaxation methods, be used whenever possible. The development of direct methods in the last decade has been one of the big breakthroughs in the development of efficient simulation codes."

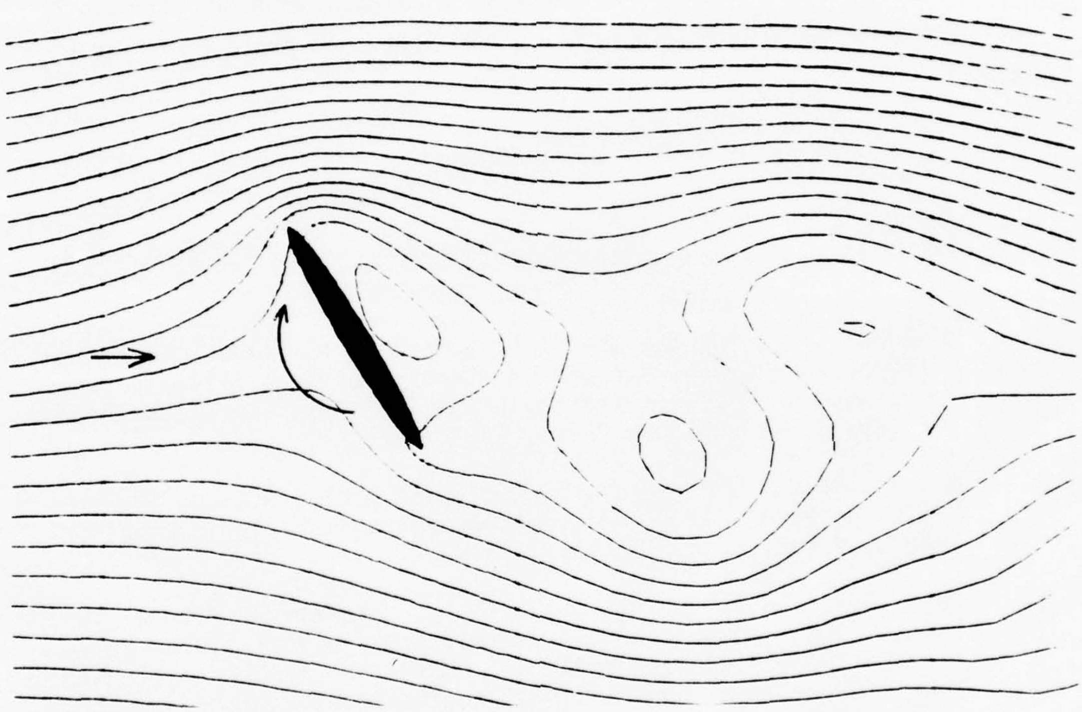


Figure 1. Streamlines Past A Rotating Plate In A Parallel Flow At A Certain Instant After The Abrupt Start Of The Plate. Re = 200, Ro = 2.

I could not agree more with this opinion. The underlying problem is this: With the flood of publications on new and improved methods it becomes increasingly more difficult to find out which methods are best for certain problems. It is impractical to expect each research group to go through the agony of programming and testing every promising new method. We at NSRDC have started this type of investigation on a very modest scale, but a more concerted effort is necessary to exploit the most advanced techniques for solving the difficult problems of hydrodynamics.

REFERENCES

1. Buzbee, B.L., G.H. Golub and C.W. Neilson, "On Direct Methods For Solving Poisson's Equations," SIAM J. Numer. Anal. 7 (1970), 627.

2. Widlund, O.B., "On the Use of Fast Methods for Separable Finite Difference Equations for the Solution of General Elliptic Problems," Sparse Matrices and their Applications, Eds., D.J. Rose, R.A. Willoughby, Plenum Press, 1972, 121.

3. Concus, P. and G.H. Golub, "Use of Fast Direct Methods for the Efficient Numerical Solution of Nonseparable Elliptic Equations," SIAM J. Numer. Anal. 10 (1973), 1103.

4. Lugt, H.J. and S. Ohring, "Efficiency of Numerical Methods in Solving the Time-Dependent Two-Dimensional Navier-Stokes Equations," Proceedings International Conference on Numerical Methods in Fluid Dynamics, University of Southampton, England, September 26-28, 1973.

5. Lugt, H.J. and S. Ohring, "Rotating Thin Elliptic Cylinder in a Parallel Viscous Fluid Flow," Fourth International Conference on Numerical Methods in Fluid Dynamics, University of Colorado, June 24-28, 1974. To be published in the series Lecture Notes in Physics by Springer-Verlag.

6. Martin, E.D. and H. Lomax, "Rapid Finite-Difference Computation of Subsonic and Transonic Aerodynamic Flows," AIAA Paper 74-11.

7. Otte, F., "Untersuchung der Stabilität einer inkompressiblen laminaren Strömung um eine ebene Platte mit und ohne Absaugung," unpublished PhD-Thesis, Institut für Luft-und Raumfahrt der Technischen Universität Berlin, 1974.

8. Buneman, O., "Vortices, the "Particles" of Fluid and Gasdynamics," Proceedings Fourth Conference on Numerical Simulation of Plasmas, Naval Research Laboratory, 1970, 440.

9. Orszag, S.A. and M. Israeli, "Numerical Simulation of Viscous Incompressible Flows," Annual Review of Fluid Mechanics, Annual Reviews, Inc., Palo Alto, California, Volume 6, 1974, 281.

NONLINEAR NUMERICAL SOLUTIONS OF TWO-DIMENSIONAL WAVE PROBLEMS

Nils Salvesen
Naval Ship Research and Development Center
Bethesda, Maryland

I would like to present some new results from our work on numerical solutions of nonlinear ship waves. It is believed that these results may be useful in the evaluation of the potential of numerical hydrodynamics in solving ship-performance problems. Before I present the results, I would like to discuss the history behind this project and why we selected this particular problem for our numerical work.

On 28 March 1972, representatives from the Office of Naval Research and the Naval Ship Research and Development Center met in order to discuss the use of numerical methods in naval hydrodynamics. It was the general opinion at this meeting that numerical methods have proven to be extremely powerful in solving fundamental fluid mechanics problems and that computer systems have advanced to such a degree of sophistication that it is now realistic to apply direct numerical methods to attack the more difficult, unsolved problems in naval hydromechanics. After this meeting a recommendation for a research program in numerical naval hydromechanics was prepared by Salvesen and Schot[1] which concludes that "the results initially most beneficial to the Navy would be achieved by concentrating on the application of numerical methods for solving nonlinear free-surface problems".

During FY74 the NSRDC started, with internal support, a small pilot program in the area of numerical solutions to nonlinear ship-wave problems. The need for expertise in theoretical and experimental hydromechanics as well as numerical mathematics was recognized and the work was conducted,

therefore, as a joint project between the Computation and Mathematics Department and the Ship Performance Department. We have found that this joint nature of the project has been very beneficial, and it is probably the main reason for the rapid progress we have made over this short time span.

In deciding on the direction for this initial project it was recognized that the nonlinear problem of a ship in a free surface is extremely difficult and in fact, there exists no complete solution to this nonlinear problem even for the most simplified body configurations. It was decided, therefore, to start with the simplified two-dimensional case and then advance to the more complicated three-dimensional case.

We would like to present nonlinear numerical results for two cases: (1) the steady-state problem of a two-dimensional disturbance below a free surface in a uniform flow and (2) the unsteady problem of an accelerating two-dimensional pressure distribution on the free surface. The numerical procedure used to solve the steady-state problem is an iterative one in which the Laplace equation is solved by finite differences in a field bounded by an assumed free-surface shape which is systematically corrected until it satisfies the nonlinear free-surface boundary condition.[2] A submerged vortex was used as the disturbance and wave resistance and free-surface profiles were computed by first- and second-order perturbation theory as well as by the complete nonlinear numerical method. The wave resistance as a function of vortex strength is shown in Figure 1. The uniform stream velocity is 10 ft/sec and the vortex submergence is 4.1 feet. Remarkable agreement between the nonlinear numerical results and the second-order perturbation results are seen. For larger disturbances one should expect some deviation from the complete nonlinear solution and the second-order perturbation solutions. In Figure 2 the wave elevations generated by a vortex of strength, $\frac{\tau}{2\pi} = -1.40 \text{ft}^2/\text{sec}$ are seen for the case of 4.5 feet submergence and 10 ft/sec uniform stream velocity. Again,

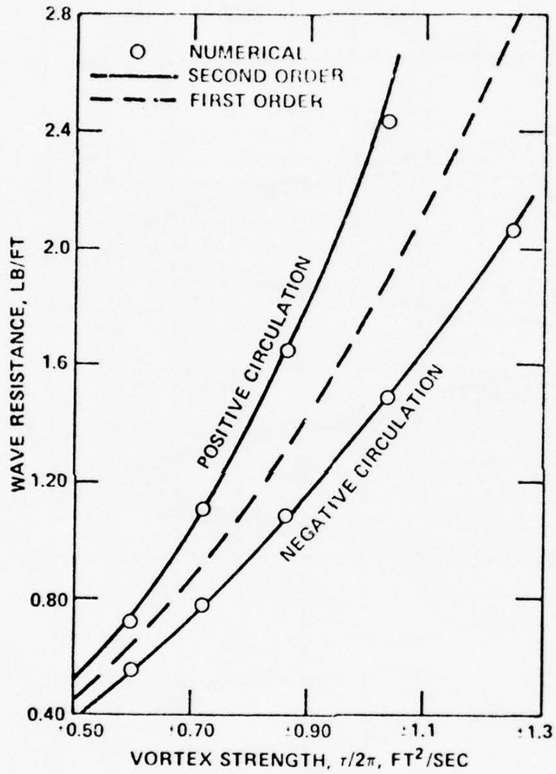


Figure 1. Wave Resistance As A Function Of Vortex Strength

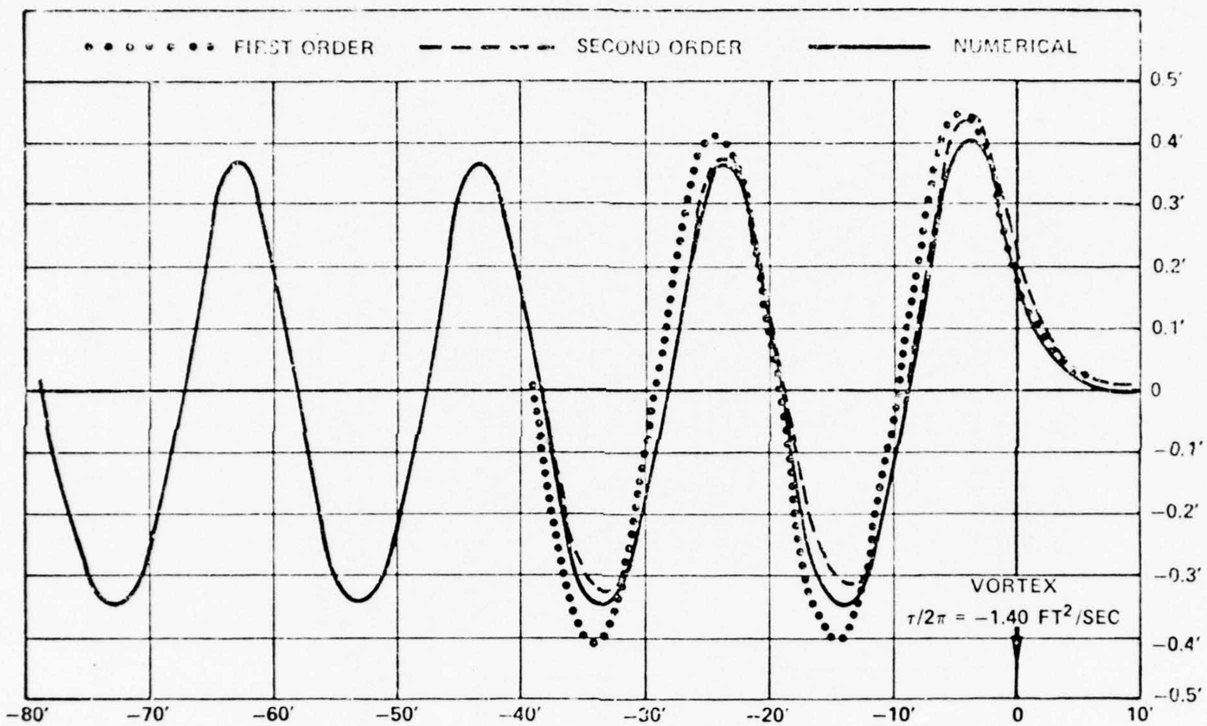


Figure 2. Wave Elevations Generated By A Submerged Vortex
In Uniform Stream

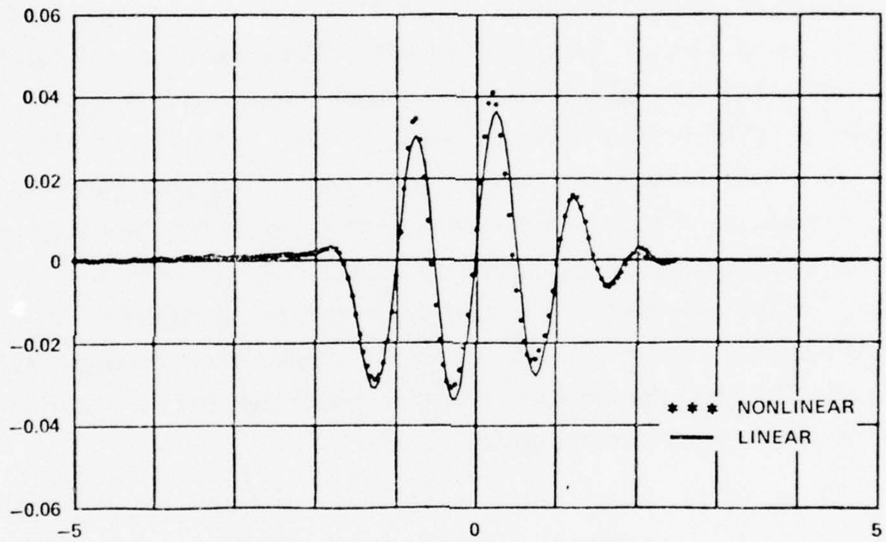


Figure 3. Comparison of Surface Elevations Generated By Linear and Nonlinear Numerical Schemes

it is seen that the nonlinear numerical wave profile agrees quite well with the profile computed by second-order perturbation theory. The good agreement between the numerical and the second-order results seems to indicate that at least for moderately steep waves, the nonlinear effects are accurately computed by this numerical scheme. Larger disturbances which produce more severe nonlinear effects will be investigated during the next year.

The unsteady pressure distribution problem was solved by a numerical method which employs a functional-expansion approach in space and finite differencing in time.[3] Numerical results have been obtained for both the linear and nonlinear equations and a comparison of wave profiles are shown in Figure 3. The pressure disturbance, in this case, is accelerated impulsively from rest in initially calm water. The flow near the disturbance achieves almost steady-state conditions quite rapidly, but thereafter the approach to steady state is slower. It is seen in Figure 3 that the wave length computed by the nonlinear method is somewhat shorter than predicted by the linear method. This difference in wavelength is in close agreement with the wavelength correction predicted by third-order perturbation theory for the steady-state case.

I think it is reasonable to conclude from these results that direct numerical methods can effectively be used in solving nonlinear two-dimensional body-wave problems and that there are potentials for extending this work to the three-dimensional ship-wave problem.

REFERENCES

1. Salvesen and Schot; "Recommendations for Advancing the Capability in Numerical Naval Hydromechanics at NSRDC," NSRDC Ship Performance Department and Computation and Mathematics Department, Report CMD-26-73.

2. von Kerczek and Salvesen, "Numerical Solutions of Two-Dimensional Nonlinear Wave Problems," ONR Tenth Symposium on Naval Hydrodynamics, MIT, June 1974.

3. Haussling and Van Eseltine, "A Combined Spectral Finite-Difference Method for Linear and Nonlinear Water Wave Problems," NSRDC Computation and Mathematics Department, Report No. 2474, August 1974.

RESEARCH ON HYDRODYNAMIC FREE SURFACES

Joel C. W. Rogers
Naval Ordnance Laboratory
White Oak, Silver Spring, Maryland

Dr. Enig has spoken to you about some hydrodynamic problems that arise in connection with the development of Naval ordnance. The purpose of my talk is to acquaint you with some of the work we have done on hydrodynamic free surfaces (water waves) to lay a foundation for the solution of these problems, and to show how this work fits into a broader perspective of tasks to be carried out.

Of course, the water does not know the mission of a body which is partly or totally submersed, and it is not surprising that many of the problems of interest to us are essentially the same, physically and mathematically, as the problems which concern the ship designer. Thus, my comments on analytic water waves will also have some bearing on the analytic water waves which Nils Salvesen and Henry Haussling have studied numerically.

We have been interested in a careful study of nonlinear water waves for two reasons: The first is that, although the comparison with experiment is always the ultimate test of our numerical work, we feel that it is nevertheless desirable to have faith in the internal consistency of our theoretical and computational work. As Professor Carrier suggested yesterday, new physical and mathematical ideas may still need development at the foundations of the theory. The second reason is that we feel that wholly new types of computational algorithms will be possible once we have sufficient understanding of the full nonlinear problem. This will be touched on later in this talk.

So far we have concentrated on water waves in inviscid fluid. In that case, if we are interested in developing solutions for a finite period of time, the essential difficulty of the problem appears when the flow is irrotational and incompressible, that is, a potential flow. In the more general case the added complication of moving vortices around is not theoretically significant.

The first viewgraph shows our progress to date on water waves in potential flow. When the initial data are analytic, we have found how to construct solutions which remain analytic for a finite period of time. By "analytic," I mean in the sense of having a Taylor expansion with a finite radius of convergence at each point. When there is a constant pressure on the free surface, and that is the minimum of the pressure on the boundary of the flow, we have found that there can be only one classical solution of the initial and boundary value problem, and that solution depends continuously on the initial data. I will return to this point further on.

In the second viewgraph, I would like to remind you that any continuous surface may be approximated with arbitrary accuracy by an analytic surface. Hence, we have really shown how to obtain solutions to the initial value problem for a "dense" class of data. The viewgraph also illustrates that such problems are profoundly nonlinear -- for example, the height of the surface need not be a single-valued function of position on a horizontal plane -- and the corresponding flows depend on the three space dimensions and time.

This is not the place to go into a proof of our results. However, we may point out that the solution of analytic partial differential equations with analytic initial data is historically associated with the names of Cauchy and Kowalewski, and that what we have done in carrying out our construction and proving its convergence is to develop a generalization of the Cauchy-Kowalewski theorem. Those of you in the audience who have

worked on similar problems in which the Laplace equation appears may not be surprised to hear that the greater part of the work has been in proving sufficiently strong regularity theorems for the Laplace equation in sufficiently general domains. In particular, we have found very precise results about the radius of convergence, at each point of an analytic surface, of the normal derivative of a solution of the Laplace equation, whose Dirichlet data on the boundary are analytic.

Some of our uniqueness and continuity results are shown in the third viewgraph. It should be observed that for most potential flows of interest to the Navy, in particular, for flows where there is a rigid bottom, $-\frac{1}{\rho}\frac{\partial P}{\partial n} > 0$ on the free surface. A slight variation of the uniqueness proof shows it to be applicable also to flows with open or closed cavities, as long as the air in the cavity is assumed to have negligible density. A further consequence of the proof is an indication of how Taylor instability may result if pressures are applied to the free surface. Professor Birkhoff has already mentioned Taylor instability. Although the introduction of surface tension appears to make the problem of Taylor instability into a well-posed problem, we feel that the singular limiting case as the surface tension vanishes is also of great interest, in much the same way that the Euler equations are an important limiting case of the Navier-Stokes equations, and that the theory of shock waves, in the limiting case as heat conduction vanishes, is important. Our uniqueness results certainly do not resolve the problem of Taylor instability in the limiting case, but they clearly bring out the preeminent influence of the pressure on the stability (in the sense of Hadamard) of the problem. There is a very strong indication that the origins of Taylor instability may lie in kinetic theory since real physical pressures can never be negative. But then, in the problem of an inverted container of liquid, there must be a positive pressure on the free surface at the bottom, or else the steady-state solution of a flat free surface with no motion in the liquid is not attainable.

On the other hand, the notion of a positive pressure exerted by a massless fluid (air) beneath the liquid, is inconsistent with kinetic theory.

The next viewgraph lists some of our plans for the near future (of the order of a year). This phase of the work is expected to be more closely related to the physics of the problem than the phase completed, and also to be directly applicable to free surface problems of the sort raised here. In the following two viewgraph we will elaborate on items (2) and (4), explaining what we mean by "weak" solutions and "efficient" algorithms. However, the main thrust of our work will be to construct solutions to the general initial value problem, not just those with analytic initial data.

In the fifth viewgraph we explain what sorts of "weak" solutions we have in mind. In particular, a treatment of the general initial value problem should be able to treat problems with splashes and sprays (without surface tension). We point out that, in the solution of these general problems, further assumptions have to be made on the air at the interface. For example, the treatment of bubble formation and collapse and related phenomena depends on whether the "air" is massless and incompressible or massless and compressible.

In regard to our concept of what constitutes an "efficient" algorithm, we indicate a strong preference for "fixed domain," as apposed to "front tracking," methods for treating some of the more pathological phenomena at the free surface. This is because our analytical investigations lead us to believe that the free surface has unbounded variation after a finite time. Such a development would almost surely doom to failure any method based only on front tracking. Mathematically, because of the importance of the Laplace equation in the water wave problem, a number of fixed domain algorithms can be formulated for somewhat simplified problems, algorithms which have a strong relation to such classical techniques as Schwarz's alternating method and Poincaré's method of balayage. As an example, although it takes us a little far afield from

the nonlinear water wave problem we are talking about principally, I show in the next viewgraph a schematic representation of a method for treating the interaction of waves on a linerized free surface with a submarine beneath the surface. The flow is conceived as evolving through a sequence of processes of wave propagation on the surface of an infinite ocean with no submersed vehicles, and wave reflection off a submarine in an infinite ocean with no free surface, with the processes alternating in time. Thus, although the flow domain itself varies in time as the submarine moves, the processes from which the flow is composed take place in fixed domains, the first in the half space $z < 0$, and the second in the domain exterior to the submarine.

My last viewgraph lists problems which would represent significant advances beyond the problems which we have already solved or plan to solve in the near future, but which are nevertheless soluble. These problems are listed in what may or may not be an order of increasing complexity, but in what is probably a natural order of solution. First, we will want to consider the presence of solid bodies breaking the free surface (ships, entering missiles, etc.). A further extension would be to account for the small but finite viscosity of water by treating also flows with vorticity, evolving under the Navier-Stokes equations. Then the pressure at the free surface is replaced by specification of surface stresses. This extension is appropriate for discussion of the generation of waves by wind. At this point, the interfacial (surface) tension of the water may reasonably be accommodated in the theory. A third development would be to try to incorporate some of the properties of the real atmosphere above the water surface, with its important effects on phenomena like cavitation and entrainment. Finally, thereis the problem of an ocean with a temperature and salinity gradient, leading to density stratification.

VUGRAPH #1

NONLINEAR WATER WAVES

PROGRESS TO DATE:

(1) ANALYTIC INITIAL DATA -- CONSTRUCTION OF SOLUTIONS ANALYTIC FOR A FINITE TIME.

(2) UNIQUENESS AND CONTINUOUS DEPENDENCE OF CLASSICAL SOLUTION ON INITIAL DATA WHEN FREE SURFACE PRESSURE IS MINIMUM OF PRESSURE (FOR EXAMPLE, ANY INCOMPRESSIBLE IRROTATIONAL FLOW IF FREE SURFACE PRESSURE IS MINIMUM OF PRESSURE ON BOUNDARY)

VUGRAPH #2

ANY SURFACE MAY BE APPROXIMATED ARBITRARILY CLOSELY BY AN ANALYTIC SURFACE.

PROBLEMS TREATED: PROFOUNDLY NONLINEAR, 3D, TIME-DEPENDENT

VUGRAPH #3

UNIQUENESS:

∂R = UNPERTURBED SURFACE

n = UNIT OUTWARD NORMAL OR ∂R

P = UNPERTURBED PRESSURE = 0 ON ∂R

Φ = UNPERTURBED VELOCITY POTENTIAL

ϕ = PERTURBATION OF VELOCITY POTENTIAL

$\frac{d}{dt}P(x,t)$ MEANS DERIVATIVE OF P AS WE FOLLOW FLUID ELEMENT AT x AT TIME t

$\zeta(x)$ FOR $x \varepsilon \partial R$ IS DISTANCE OF PERTURBED SURFACE FROM ∂R, MEASURED ALONG n

$$\frac{d}{dt} \quad \int_R (\nabla\phi)^2 dV + \int_{\partial R} (\zeta(x))^2 \; \left(-\frac{1}{\rho}\frac{P}{\partial n}\right) dS$$

$$= \lim_{\xi \to 0} \frac{1}{\xi} \int_{2R} (\zeta(x))^2 \, \frac{1}{\rho} \frac{d}{dt} \, P(x - n\xi, t) \, dS$$

$$- 2\int_R \sum_{i,j} \frac{\partial\phi}{\partial xi} \; \frac{\partial\phi}{\partial xj} \; \frac{\partial^2\Phi}{\partial xi \partial xj} \; dV$$

$P \neq 0$ ON FREE SURFACE: MAY GET TAYLOR INSTABILITY

ORIGINS OF TAYLOR INSTABILITY MAY LIE IN KINETIC THEORY

VUGRAPH #4

NONLINEAR WATER WAVES

PLANS:

(1) FIND SOLUTION TO GENERAL INITIAL VALUE PROBLEM

(2) IDENTIFY UNIQUE ACCEPTABLE WEAK SOLUTION

(3) GIVE SOME OF ITS PROPERTIES

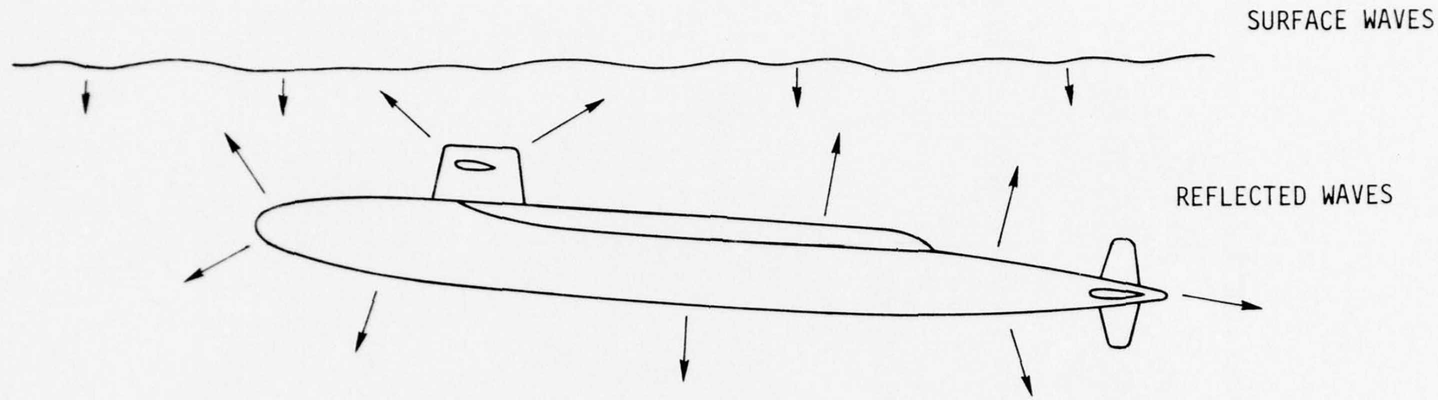
(4) DEVELOP EFFICIENT ALGORITHM FOR SOLVING WATER WAVE PROBLEM.

FULL TREATMENT OF WATER WAVE PROBLEM SHOULD HANDLE SPLASHES AND SPRAYS.

ASSUMPTIONS ON "AIR" ABOVE AFFECT NOTION OF ACCEPTABLE WEAK SOLUTIONS.

"EFFICIENT" ALGORITHMS: FOR EXAMPLE, FIXED DOMAIN METHODS, CF, ALTERNATING METHOD OF SCHWARTZ, POINCARE'S METHOD OF BALAYAGE.

METHOD OF REFLECTED WAVES



1. WAVES TRAVEL BETWEEN SURFACE AND SUBMARINE IN TIME Δt.
2. FLOW DETERMINED BY SOURCES ON SURFACE OF SUBMARINE AND TRAVELING WAVES ON FREE SURFACE.
3. MOTION OF SUBMARINE DETERMINED BY THRUST AND PRESSURE FORCES.
4. SOLUTION MARCHES IN TIME IN INCREMENTS AT Δt.

VUGRAPH #6

VUGRAPH #7

PROBLEMS OF INCREASING COMPLEXITY:

1. PRESENCE OF SOLID BODIES BREAKING SURFACE
2. VISCOSITY, SURFACE STRESSES, SURFACE TENSION
3. REAL FLUID ABOVE SURFACE
4. STRATIFIED OCEAN

REPORT OF THE
WORKSHOP REVIEW COMMITTEE

The Workshop Review Committee was assigned the task of assessing the results of the proceedings, arriving at conclusions bearing on potential advances in numerical hydrodynamics, and recommending those actions best suited to realizing such advances. The results of the Committee's review is summarized in the ten paragraphs to follow.

1. This well-attended Workshop in four sessions, organized by the National Academy of Sciences-National Research Council and the Office of Naval Research, featured talks and discussions by personnel both from within the Navy, associated currently with Navy programs in numerical hydrodynamics, and highly qualified and experienced personnel involved with non-Navy programs, such as those of the Atomic Energy Commission, National Aeronautics and Space Administration, Advanced Research Projects Agency and the National Center for Atmospheric Research. The Workshop provided a good, if not overly detailed, view of the current status of the field and of future possibilities of importance to the Navy. Some important conclusions can be drawn and recommendations made.

2. Capability in numerical hydrodynamics is still growing rapidly in terms of technique (see Session II), machines and cost reduction (see Session III), and it is estimated that this growth will continue for another decade, resulting in at least one and perhaps two orders of magnitude reduction in unit cost of computations.

3. During the past decade, capability was limited and largely restricted in application to relatively simple problems in two space dimensions, but including unsteadiness in time; exceptions are homogeneous turbulence in three space dimensions at low Reynolds numbers, and the beginning of three dimensional transonic wing calculations. As a result, numerical hydrodynamics has not yet played a very important

role in applications. During the next decade applications of numerical hydrodynamics will be extended more widely to viscous problems in three space dimensions, but simulation of turbulence will still be limited to relatively low Reynolds numbers. Only in further decades is it likely that high Reynolds number flows of any complexity will be treated adequately.

4. The successful treatment of turbulent flows at practical scales by numerical hydrodynamic methods would be of great importance for both research and engineering practice as most turbulent flows cannot at all be adequately treated by analytical means at present, nor is it likely that such means of general applicability will be available in the foreseeable future.

5. In application to non-turbulent flows, numerical hydrodynamics methods must often compete with simpler computational methods utilizing a variety of analytical techniques and results. Nevertheless, there are probably important classes of problems for which numerical hydrodynamics should and will become the method of choice in the decades ahead, as techniques improve and computing costs are further reduced. These problems are not yet clearly identifiable, but will probably include those with non-linear and/or complicated boundary conditions and boundaries, highly unsteady flows, and flows in non-homogeneous media. These include several problems of practical importance to the Navy, particularly those involving heavy wave-making, spray generation, water impact, cavitation, vortex wakes, internal wave generation, and complicated surface wave phenomena. For example, in connection with the development of high performance vehicles, such as SES and hydrofoils, there would seem to be important prospects for the eventual use of numerical hydrodynamics methods in design.

6. The successful application of numerical hydrodynamics methods to practically important hydrodynamic problems will require a persistent

long range effort by skilled and experienced persons who will have to develop necessary mathematical techniques, both for turbulent and non-turbulent flows. Ideally this effort should match improvements in computing machine technology. It is questionable whether the current effort is doing this or is sufficiently coherent to fully realize the gains which are potentially available.

7. The Office of Naval Research is well-advised to consider a specific coherent program to aid in the continual development during the next decade of numerical hydrodynamics techniques of importance for the solution of hydrodynamic problems. This program would ideally make use of the best experience and the best personnel now working in the subject, and would be broadly based enough to encompass a variety of projects and viewpoints. Further, an adequate working contact should be ensured between experimentalists, analysts, ultimate users, and those engaged in numerical hydrodynamics development. With regard to size, the organization of a large scale "crash" effort does not seem warranted at the present time, but the program should be of adequate size to ensure steady progress in technique and applications, matching growth in machine capability. Above all, the program should be of a sufficiently long range nature.

8. The long range goals (10-20 years) of such a program should include the development of methods to deal with practical turbulent flows (such as the separated flow about the stern of a ship or turbulent wakes in a stratified ocean), and complex cavitating flows (such as cavitation inception and partial cavitation or a ship's propeller or high-speed water-jet inlet), taking into account both the formidable nature of these problems and the gains to be realized in their solution.

9. The short and medium range goals (3-10 years) should include applications to a few complex but important problems of general interest in a naval context. These might include: (a) the "exact" inviscid (potential) flow about a real displacement ship making waves; (b) the

"exact" inviscid flow about a deeply submerged submarine including the vortex wake behind the control surfaces; (c) the "exact" inviscid unsteady flow about a three-dimentional body entering water (impact). Such applications should not, however, be to the exclusion of others involving more complicated physics, such as in the case of cavitation problems and flows in non-homogeneous media, for in the long run these might well prove to benefit most from numerical hydrodynamic techniques.

10. As at present, the Office of Naval Research should continue its interest in the application of existing numerical hydrodynamic techniques by encouraging hydrodynamic research which utilizes numerical hydrodynamics as a tool when possible and advisable, and by taking steps to aid in the dissemination of results and techniques through specialists meeting or through already established means such as the Symposium on Naval Hydrodynamics.

INVITED GUESTS

A. H. Aitken, NRL

J. Bai, MIT

G. Baker, Harvard

A. Berman, NRL

J. P. Boris, NRL

L. D. Bram, ONR

J. P. Breslin, Stevens Inst

S. L. Brodsky, ONR

J. H. Brown, NUC

J. Buck, NAVSEC

S. Burnstein, Mathematical Applications Group, Inc.

V. Castelli, Columbia Univ

R. Chan, Space Applications, Inc.

S. I. Cheng, Princeton

G. Christoph, NUSC

M. Ciment, NOL

M. Cooper, ONR

R. Cooper, ONR

R. T. Davis, Univ of Cincinnati

S. W. Doroff, ONR

V. Dougalis, Harvard

B. E. Drimmer, NAVORD

H. Elrod, ONR Branch Office

J. W. Enig, NOL

F. H. Featherston, CAPT, USN, ONR

J. Frye, NUSC

P. Garabedian, New York Univ

J. T. Geary, CAPT, USN, NRL

G. H. Gleissner, NSRDC

J. W. Granet, NAVSHIP

R. V. Hayes, CAPT, USN, ONR

A. V. Hershey, NWL

J. L. Hess, Douglas Aircraft Co.

J. D. Hwang, Ames Research Center

D. Jansson, ONR

F. C. Jones, RADM, USN, NMAT

T. Chin Jones, ACDA

H. Keller, California Institute of Technology

R. W. King, RADM, USN, NAVSHIP

W. S. King, Rand Corp

C. Knight, Harvard

R. L. Lau, ONR Branch Office

J. S. Lawson, Jr., NMAT

P. Lax, New York Univ

A. Leonard, Stanford Univ

P. Lieber, Univ of California, Berkeley

A. Loeb, Franklin Inst

D. Lombard, SUBCOM

R. J. Lundegard, ONR

P. Mandel, MIT

R. W. Manning, NAVSHIP

S. R. Marcus, NAVORD

C. Morawetz, New York Univ

G. Moretti, Polytechnic Inst of New York

D. Nelson, NUC

P. W. Nelson, CAPT, USN, NSRDC

B. Nichols, Los Alamos National Labs

G. Nicoles, Naval Ordnance Station

W. Noh, Lawrence Livermore Lab

T. F. Ogilvie, Univ of Michigan

B. S. Orleans, NAVSHIP

S. Patterson, National Center for Atmospheric Research

T. E. Peirce, NAVORD

J. F. Polk, BRL

A. Powell, NSRDC

S. G. Reed, Jr., ONR

P. Roache, Science Applications, Inc.

G. O. Roberts, Northwestern Univ

J. C. W. Rogers, NOL

M. Rogers, Air Force Office of Scientific Research

N. Salvesen, NSRDC

T. Sarpkaya, NPGS

E. A. Schroeder, NSRDC

S. F. Shen, Cornell Univ

J. M. Solomon, NOL

D. C. Tolefson, Newport News Ship Building

J. Toomre, NRL

M. D. Van Orden, RADM, USN, ONR

A. H. Van Tuyl, NOL

J. Viecelli, Lawrence Livermore Lab

C. Von Kerczek, Johns Hopkins Univ

W. C. Webster, Univ of California, Berkeley

J. V. Wehausen, Univ of California, Berkeley

K. S. Wen, Univ of California, Berkeley

W. A. Whitaker, LTC, ODDR&E

M. Wilkins, Lawrence Livermore Lab

R. E. Wilson, NOL

R. Wai-Chung Yeung, MIT

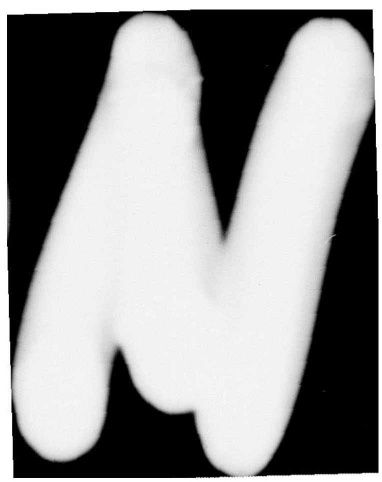
C. A. Zoltani, BRL

END

DATE
FILMED

9-77

DDC



AD-A043 448 NATIONAL ACADEMY OF SCIENCES-NATIONAL RESEARCH COUNCI--ETC F/G 20/4
PROCEEDINGS OF THE WORKSHOP ON NUMERICAL HYDRODYNAMICS.(U)
JUL 75 N00014-67-A-0244-0013

UNCLASSIFIED NL

5 of 5
AD A
043448

SUPPLEMENTARY

INFORMATION

END
DATE
FILMED
7-80
DTIC

AD A
043

SUPPLEMENTARY

INFORMATION

AD-A043 448

ERRATA

Proceedings of the Workshop on Numerical Hydrodynamics

Page 3, Paragraph 1, Line 7.

For ....apologies of our....

Read ....apologies and those of our....